Journal of
MECHANICAL and ENERGY
ENGINEERING

ISSN: 2544-0780 | e-ISSN: 2544-1671
Vol. 6(46) | No. 1 | June 2022 | pp. 7-12
DOI: 10.30464/jmee.2021.6.1.7

PREDICTION OF CAVITATION INSIDE MINI-SAC DIESEL INJECTOR NOZZLE USING COMPUTATIONAL FLUID DYNAMICS

Vaibhav BANSODE[1*], Munna VERMA[2], Amar PANDHARE[3], Sandip SHINDE[4]

[1*] Mechanical Engineering Department, STES's Smt. Kashibai Navale College of Engineering, Pune-41, Maharashtra, India, e-mail: bansode.vaibhav@gmail.com
[2] Mechanical Engineering Department, Bhagwant University, Ajmer, Rajasthan, India
[3] Mechanical Engineering Department, STES's Sinhgad College of Engineering, Pune-41, Maharashtra, India
[4] Mechanical Engineering Department, STES's Smt. Kashibai Navale College of Engineering, Pune-41, Maharashtra, India

(Received 16 February 2022, Accepted 2 March 2022)

Abstract: The performance of the internal flow of the fuel injector is impeded by several factors. The nozzle is one of the factors, being typically about a millimeter long and a fraction of a millimeter in diameter. Cavitation inside the diesel injector nozzle is associated with local pressure distribution. At flow areas with sharp corners, the pressure may locally drop below vapour pressure. The aim of this study is to assess the impact of turbulence and cavitation models on the prediction of flow in diesel injection nozzle. In the present study, an analysis of an existing 6 hole mini-sac diesel injector nozzle is carried out using a CFD package. The main objective of the research is to design a nozzle to avoid cavitation and to find out the contribution of different parameters through parametric study. Cavitation is a complex phenomenon whose appearance depends on the physical as well as flow properties of the flowing substance. Thus, for a better visualization of cavitation, a 3D CFD simulation of mini-sac injector nozzle is carried out. An analysis of a single nozzle hole of a mini-sac diesel injector nozzle is considered for the analysis, as the flow is uniformly distributed through each nozzle. As the three-dimensional geometry of mini-sac nozzle is complicated, therefore tet/hybrid element with T-Grid meshing scheme is used, for good surface meshing. The analysis is carried out at injection pressure of 500 bar. The CFD results are validated against test data with the maximum deviation for the mass flow rate of 8.67% at full needle lift.

Keywords: cavitations, mini-sac diesel injector nozzle, needle lift, parametric study, CFD.

1. INTRODUCTION

The injection nozzle is the most important for engineering applications relating to the fuel injection system. The inner flow through an injection nozzle is a very important factor for spray formation and atomization generated by the nozzle, and it is related to the combustion potency and the resultant formation of pollutants. An intensive understanding of the complicated nature of flow into the nozzle and flow contagious to the nozzle is critical for predicting spray development [1]. In turn, cavitation occurs within the diesel nozzle because of a pressure drop below the saturated vapor pressure that is additionally known to attenuate the combustion efficiency directly and indirectly.

Computational Fluid Dynamics has proved its importance in recent years; this is a simulation tool for modeling thermal and fluid flow problems [2]. Fuel injection duration and injection pressure do not play any great role in the dynamic formation of the varying cavitation patterns [3]. The CFD model provides information about the local flow field at the location of the vortex. Vortex cavitation is a source of a cycle to cycle and spray to spray variation in diesel fuel injectors [4].

Cavitation and gas entrapment inside the injector nozzle orifice can greatly affect the local and global behavior of the flow. For a given complicated nature of the nozzle, several investigations have focused on the internal nozzle flow. Once the nozzle flow is understood, spray breakup may become a more tractable problem. However, the details of the flow through the nozzle are quite complicated and have become a field of study on their own. The study of the internal nozzle flow includes turbulence, developing pipe flow and cavitation [5]. The ultimate goal of the nozzle is to produce spray and so our current approach begins with understanding the spray. The present study focused on the development of a CFD model for cavitation.

Numerical analysis of minisac diesel injector nozzle is performed using FLUENT 6.3.26 CFD package to study vapour distribution along nozzle hole and amount of vapor fraction for given pressure range.

For a different needle lift, mass flow rate at the outlet is calculated for a steady state condition and validated against test data. Finally, a CFD analysis of minisac nozzle for unsteady condition is done and parametric study is carried out for the selection of an optimum combination of geometrical parameters to minimize cavitation.

2. THEORETICAL BACKGROUND

In the present analysis, a six hole nozzle with sac volume is considered with all the dimensions in mm, and it is shown in Figure 1. The nozzle is consisting of six holes and each hole located axe-symmetrically. Therefore, only a single nozzle hole is considered for the analysis to minimize the computational efforts.

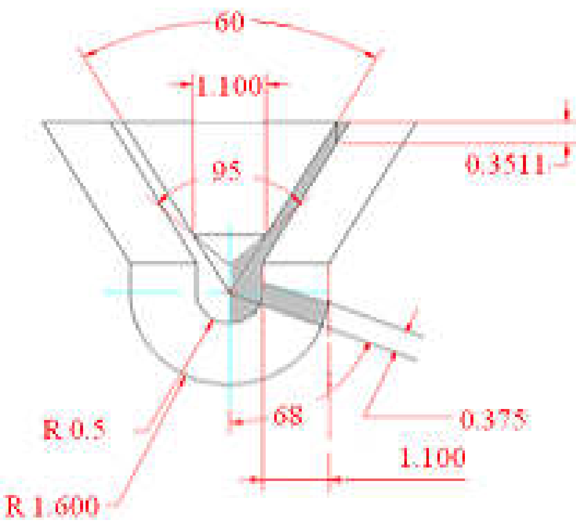

Fig. 1. Geometry of mini-sac nozzle

3. ANALYSIS

The analysis is carried out analytically and numerically. The analytical solution is used to calculate the reference mass flow rate at the outlet, and the numerical solution provides the vapour fraction along the nozzle hole.

3.1. Analytical solution

The analytical calculation is done with a simple nozzle flow. Therefore, the reference mass flow Mref is calculated using the Bernoulli equation. The value of the coefficient of discharge is calculated from the correlation reported by Martynov [1].

1. Value of *Cd*.

The value of the coefficient of discharge is calculated from the cavitation number. P2 and P1 are inlet and outlet pressures respectively.

$$CN = \frac{(p_2-p_1)}{(p_1-p_v)} = \frac{500-5}{5-0.054} = 99.98. \quad (1)$$

2. Velocity at the outlet.

Pressure drop:

$$\Delta P = 495 \times 105 \text{ Pa}. \quad (2)$$

Coefficient of discharge is equal to:

$$Cd = 0.67, \quad (3)$$

$$\rho = 832 \text{ Kg/m}^2, \quad (4)$$

$$A = 0.11046 \text{ mm}^2, \quad (5)$$

$$V = Cd\sqrt{\frac{2\times\Delta P}{\rho}} = 0.67 \times \sqrt{\frac{2\times495\times10^5}{832}} = 231.116\frac{\text{m}}{\text{s}}. \quad (6)$$

3. Mass flow rate at the outlet.

Mass flow rate is equal to:

$$\rho AV = 832 \times 0.11046 \times \times 10^{-6} \times 231.116 = 0.0212 \text{ Kg/s}. \quad (7)$$

3.2. CFD solution

The analysis is done using the CFD code. The steps in the analysis include geometry creation, meshing, assigning boundary conditions, a solution and post-processing. In the geometry creation and meshing, the nozzle geometry is modelled and divided in small elements with an interval size of 0.8. After that, assigning boundary conditions and domain defining is carried out. Then, the system is imported to a suitable solver to solve it using a different physical model. In post-processing, various parameters are predicted to obtain more details about the results. The present system is analysed using the abovementioned steps with a proper model as follows.

3.3. Computational model

To analyse the flow characteristics of the in-nozzle flow, the 3D axisymmetric single nozzle is considered as a continuum. The nozzle was modelled using GAMBIT 2.3.16 and meshing was done with a tet/hybrid element and a T-Grid meshing scheme for good surface meshing. As the dimensions of the nozzle are in millimetres, the interval size was selected to be 0.8. The meshed geometry is as shown in Figure 2.

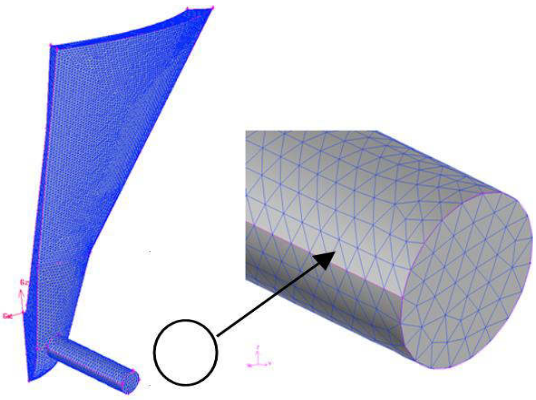

Fig. 2. Meshing of Mini-sac Nozzle

Initial and boundary conditions

In the present analysis, the pressures at the inlet and at the outlet are specified with the pressures of 500 and 5 bar respectively with a temperature of 300 K and a vaporization pressure of 5400 Pa. The density and viscosity of diesel is 832 kg/m^3 and 0.0065. The commercial code consists of different solver models such as turbulence, radiation and phase change. In the present analysis, the reference velocity indicates higher turbulent flows. Therefore, a standard *K-ε* model was employed in the analysis [4]. The boundary assigned conditions are shown in Figure 3. Then model is imported to a fluent with assigned boundary conditions.

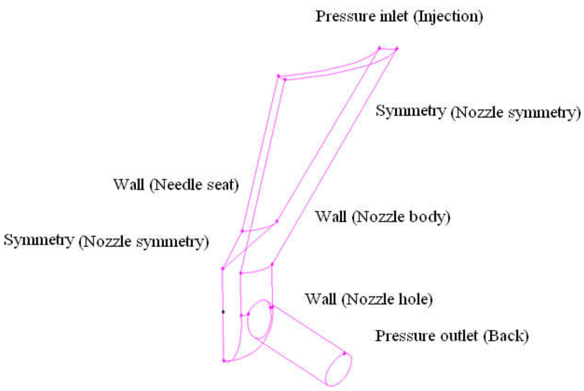

Fig. 3. Boundary Conditions Assigned to Nozzle

Results and discussion

The performance prediction for the mass flow rate with different needle lifts at a steady state condition is carried out. The variation of the needle lift Vs mass flow rate is presented in Figure 4. It is observed over the full needle lift in the maximum variation in the predicted and experimental results, and it is 8.67%.

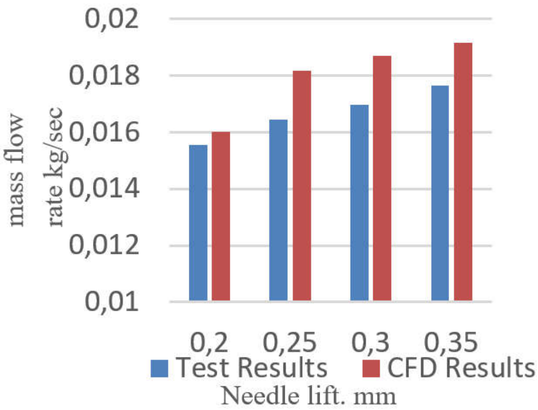

Fig. 4. Variation of needle lift Vs mass flow rate

The pressure and vapour distribution along the nozzle length is presented in Figure 5 and 6. The cavitation formation is obtained at the solid boundary of the nozzle hole.

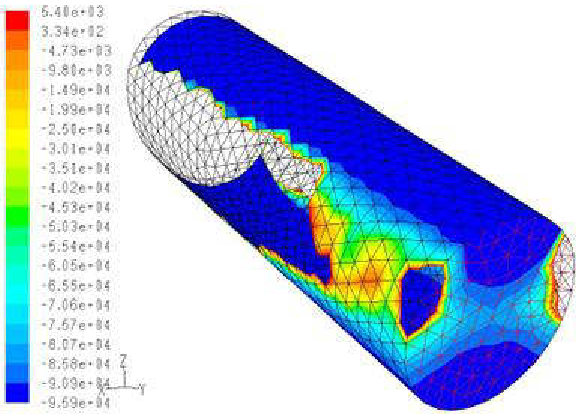

Fig. 5. Pressure (in Pa) contour plot

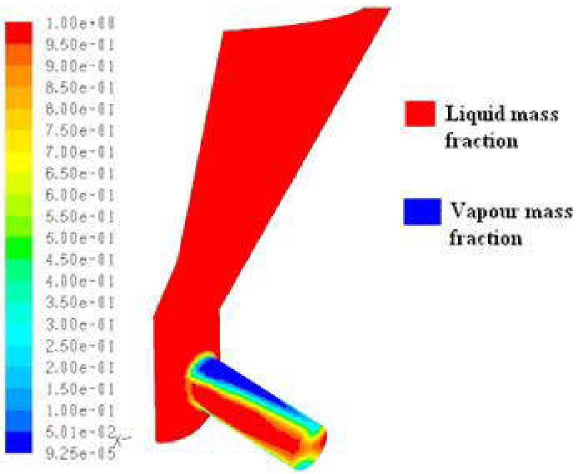

Fig. 6. Phase contour plot

Parametric study

As cavitation occurs along the nozzle hole, there is need to study the influence of different parameters. There are different geometrical parameters considered for parametric study such as the sac volume, the nozzle hole angle and the corner radius to predict the cavitation (Fig.7 to 12).

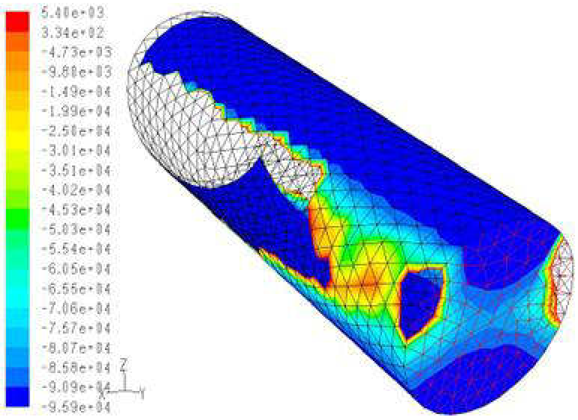

Fig. 7. Pressure (in Pa.) contours for 0.5 mm, 0 mm & 68°

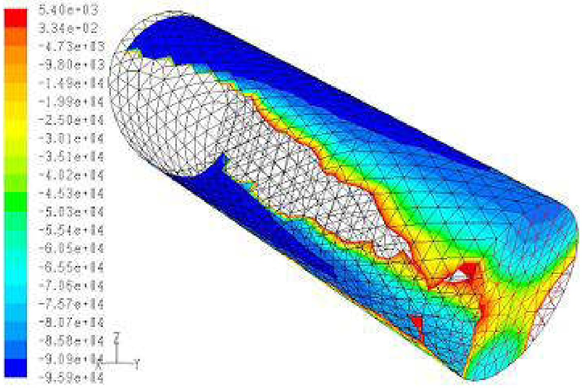

Fig. 8. Pressure (in Pa.) contours for 0.475 mm, 0 mm & 68°

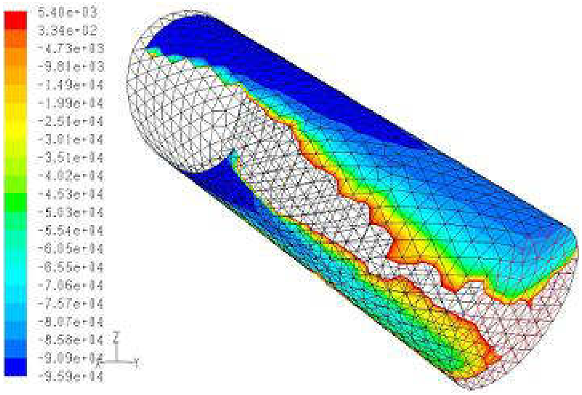

Fig. 9. Pressure (in Pa.) contours for 0.475 mm, 0 mm & 65°

Figures indicate that the formation of cavitation mostly along the nozzle hole length near the solid boundary is due to abrupt changes in the flow path while the fuel travels from the sac area to the nozzle hole inlet. In the figure, the nomenclature of the first fraction and of the second fraction corresponds to the sac volume radius and the corner radius respectively. The last number indicates the nozzle hole angle.

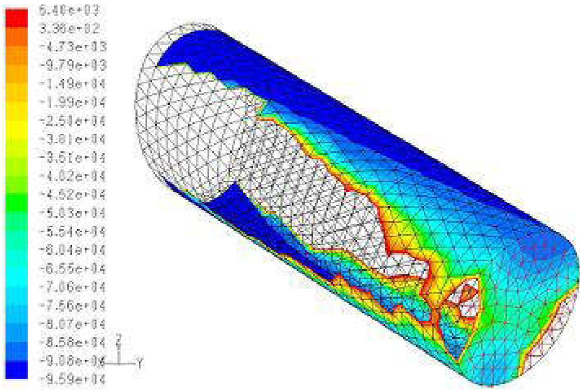

Fig. 10. Pressure (in Pa.) contours for 0.5 mm, 0 mm & 62°

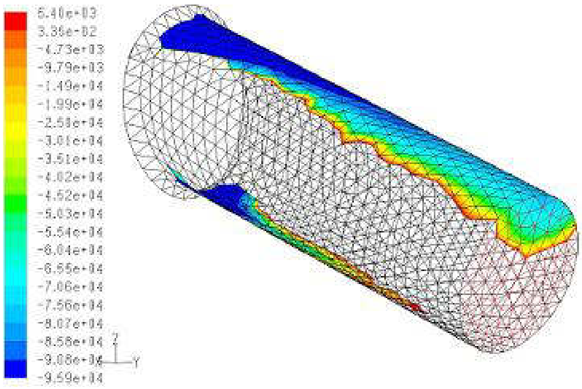

Fig. 11. Pressure (in Pa.) contours for 0.5 mm, 0.375 mm & 65°

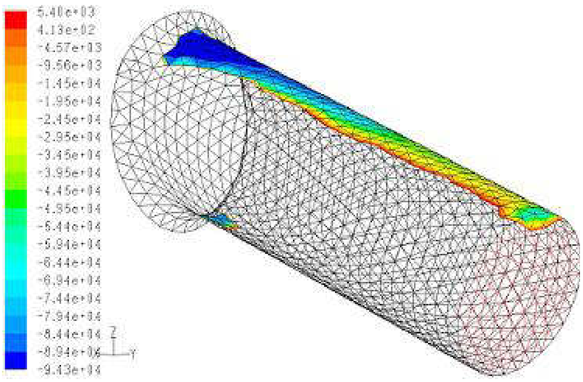

Fig. 12. Pressure (in Pa.) contours for 0.5 mm, 0.75 mm & 65°

Figure 13 shows the graph of a combined effect of the parameters on cavitation. It is useful to select the combination with minimum cavitation.

From the graph, it is evident that the corner radius plays an important role to minimize cavitation. From the viewpoint of the nozzle hole angle, it is difficult to predict the role of the nozzle hole angle for different sac volumes and different corner radiuses.

4. TEST RESULTS

For the injector nozzle, both Charged Coupled Device (CCD) images and Laser Doppler

Velocimetry (LDV) measurements are used. The velocity obtained at different vertical planes is shown in Figure 14. The mean axial velocity component in the direction of the hole axis was measured. The results were presented for the flow of fully developed cavitation. The plotted velocity was normalized with the mean injection velocity ($U_{injection}$) derived from the flow rate and the hole cross-sectional area. The computational results were presented at respective locations. Test cases are shown in Table 1.

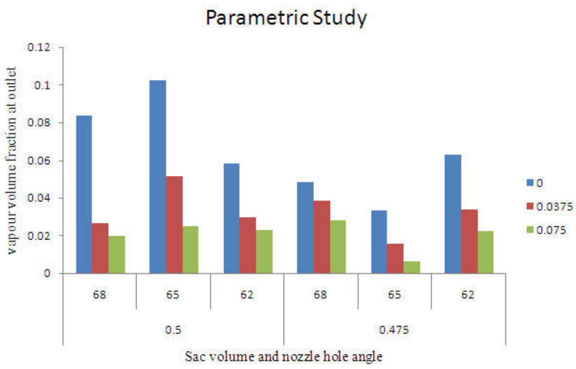

Fig. 13. Comparison of different combinations of parameter for minimizing of cavitation

Tab. 1. Test Cases Used for Validation

Large scale	Cavitation No.	$P_{injection}$ (bar)	$U_{injection}$ (m/s)	Flow regime
	1.49	3.0	16.0	Fully developed

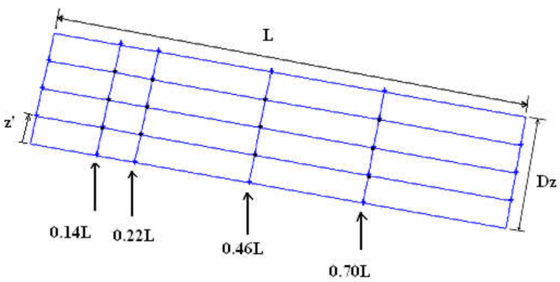

Fig. 14. Numerical Grid and LDV Measurement Planes for Velocity Calculation along Nozzle Hole Length

5. CONCLUSIONS

The methodology is extended to predict the performance of a given mini-sac diesel injector nozzle and it is assessed against both test and analytical results. Also, a parametric study was performed to predict the performance of a given mini-sac diesel injector nozzle over a range of geometrical parameters.

The following conclusions are made based on the CFD results obtained:

1. A larger sac volume has always a problem of dribbling; therefore, from that point of view, the radius of the sac volume is reduced to 5%, and it shows an effect of cavitation. Figures 6a and 6b indicate that less cavitation occurred at the minimum sac volume area.
2. In the analysis, the nozzle hole angle was reduced up to 10% and its effect on cavitation was analysed. It is difficult predict the effect of the nozzle hole angle on cavitation. However, it was observed that the minimum cavitation occurred for the nozzle cone angle of 65°, which is quite obvious from the comparisons of Figures 6b, 6c and 6d.
3. In the case of the corner radius, the corner radius was increased in the relation to the nozzle hole diameter (CR = 0, 0.1 and 0.2 D) and the effect of the corner radius is analysed in Figures 6d, 6e and 6f. It indicates that the role of the corner radius is more dominant for minimizing cavitation.
4. For given geometry and boundary conditions, the 65° nozzle hole angle gives minimum cavitation for the sac volume of 0.475 mm and maximum cavitation for 0.5 mm with the corner radius.
5. The minimum vapour fraction is observed for a combination of the 0.475 mm sac volume with the 65° hole angle and the 0.075 mm corner radius.

6. FUTURE SCOPE

The present work can be extended to study the effect of variation in the nozzle hole diameter and the nozzle hole length. In the present study, an analysis is carried out with fixed needle lift. It may possible to analyze the cavitation inside the nozzle with transient needle lift.

Acknowledgements

Part of this work was supported by the Mahindra and Mahindra Group. The Authorwould like to thank Mr Shrinath Korgaonkar from Walchand College of Engineering Sangli. Special thanks to Dr. B. S. Gawali, Professor, Walchand College of Engineering Sangli.

Nomenclature

Acronyms

CFD – Computational Fluid Dynamics
LDV – Laser Doppler Velocimetry
CCD – Charged Coupled Device

References

1. Sergey Martynov "Numerical simulation of cavitation process in diesel injectors", PhD Thesis, University of Brighton, 2005.
2. M. Gavaises, E. Giannadakis, "Modeling of cavitation in large scale diesel injector nozzle". In ILASS 2004. Nottingum, UK.2004.
3. H. Roth, K. Omae *et. al*, "effect of multiple injection strategy on the nozzle hole cavitation". SAE Paper 2005-02-1237.
4. M. Gavaises and A. Andriotis, "cavitation inside multi-hole injectors for large diesel engines and its effect on the near nozzle spray structure". SAE Paper 2006-01- 1114, 2006.
5. M. Gavaises and A. Andriotis "Vortex flow and cavitation in diesel injector nozzles" J. Fluid Mech. 2008 vol. 610, pp. 195–215.
6. D. Papoulis, C. Arcoumanis *et. al*, "Evaluation of predictive capability of diesel nozzles cavitation models" SAE Paper 2007-01-0245, 2007.
7. C. E. Brennen "Cavitation and bubble dynamics", Oxford University Press. 1995.
8. Heywood, John "Internal Combustion Engine Fundamentals, McGraw-Hill, New York, 1989.
9. H.K.Versteeg and W. Malalaserekha "An Introduction to computational fluid dynamics", Publication Longman scientific and technical, Inc.
10. R. S. Lagumbay, Jin Wang (2005), "Numerical simulation of supersonic 3- phase cavitating jet flow through a gaseous medium in injection nozzle", IMECE2005-82948, 5-11 Nov.
11. H. Roth, M.Gavaises and C. Arcoumanis "Cavitation initiation and its development and link with flow turbulence in diesel injector nozzle." SAE Paper 2002-01-0214, 2002.
12. S. B. Martynov and D. J. Mason "Numerical simulation of cavitating flows based on hydrodynamic similarity". IMechE 2006.
13. L. C. Ganippa, G. Bark *et. al*. "Comparison of Cavitation Phenomena in Transparent scaled-up Single-Hole Diesel Nozzles" CAV2001:sessionA9.005
14. Henry Weller and Niklas Nordin, "Modeling injector flow including cavitation effects for diesel applications" FEDSM2007-37518.
15. E. Giannadakis, M. Gavaises And C. Arcoumanis, "Modelling of cavitation in diesel injector nozzles", J. Fluid Mech. (2008), vol. 616, pp. 153‑193.
16. Lian Duan , Shou-qi Yuan *et.al*, "Injection performance and cavitation analysis of an advanced 250 MPa common rail diesel injector", International Journal of Heat and Mass Transfer 93 (2016) 388–397.
17. Xinping Long, Qi Liu, Bin Ji , Yiyuan Lu, "Numerical investigation of two typical cavitation shedding dynamics flow in liquid hydrogen with thermodynamic effects", International Journal of Heat and Mass Transfer 109 (2017) 879–893.
18. Liang Zhang, Zhixia He *et. al*., "Simulations on the cavitating flow and corresponding risk of erosion in diesel injector nozzles with double array holes", International Journal of Heat and Mass Transfer 124 (2018) 900–911

Biographical notes

Vaibhav Bansode received his M.Tech. degree in Heat and Power Engineering from Walchand College of Engineering Sangli, Maharashtra in 2010. Since 2010 he has been working as an Assistant Professor in the Department of Mechanical Engineering at the STES's Smt. Kashibai Navale College of Engineering, Pune-41. His scientific interests focus on Thermodynamics, Fluid Mechanics, Internal Combustion Engines, Heat Transfer, Computational Fluid Dynamics. He has participated in several international and national conferences, workshops and seminars. He has published papers in 10 international and 6 national journals and conferences.

Munna Verma is working as a Head of Department, Bhagwant University, Ajmer. He has received his B. Tech degree in Mechanical Engineering from UIET, CSJM University, Kanpur, India. Further studied in Mechanical Engineering and received his M. Tech (Thermal Engg.) from National Institute of Technology Patna, India and also completed Ph.D. from National Institute of Technology, Patna, India on the topic Investigation on Fluidized Bed Drying and Co- firing of Biomass under the guidance of Dr. A.N. Sinha (Professor & Head, ME, NITP) and Dr. C. Loha (Senior Scientist, Thermal Engg. Division, CSIR-Central Mechanical Engineering Research Institute, Durgapur). . His scientific interests focus on Heat and Mass Transfer, Bio-energy, Experimental Fluid Mechanics, Turbulence, Flow Visualization, CFD.

Amar Pandhare received his Ph.D from Sinhgad College of Engineering Maharashtra in 2014. He is currently working as a HoD Mechanical , Sinhgad College of engineering, Pune.His Scientific interests focus on Thermodynamics, Fluid Mechanics, Internal Combustion Engines, Heat Transfer, Artificial Intelligence and Machine Learning. He has participated in several international and national conferences, workshops, webinars and seminars. He has published papers in various international and national journals and conferences.

Sandip Shinde is working as an Assistant Professor in the Department of Mechanical Engineering at the STES's Smt. Kashibai Navale College of Engineering, Pune, since 2012. He has completed his bachelor degree in Mechanical Engineering and post graduation in ME Heat Power from Savitribai Phule Pune University. His scientific interest focuses on Fluid Mechanics, Heat Transfer, IC Engines and Turbo Machines. He has participated in several international and national conferences, workshops, webinars and seminars. He has published papers in various international and national journals and conferences.