CFD ANALYSIS IN COAXIAL INJECTOR OF A CRYOGENIC ROCKET ENGINE AND EVALUATION OF ATOMISATION CHARACTERISTICS USING SIMULANT FLUIDS

S. Sujith* and N.V. Mahalakshmi**

Abstract

A Study of atomisation and mixing of 2-phase propellants using simulant fluids injected from a coaxial injector of an operating cryogenic rocket engine is carried out and presented along with the CFD analysis of flow within it. The original injector jet (Jet 3) designed, modeled and analysed has swirl for LOX (water) alone with GH_2 (air) having no swirl. Injector jet having inner casing and outer sleeve is modeled using I-DEAS software and analysed for inner core fluid domain (liquid) and annular fluid (air) domain. Results of flow analysis inside jet revealed asymmetric nature of air flow at exit due to the air fluid domain itself is not axisymmetry in the original jet. This non-axisymmetry of 2-phase fluids at the exit of injector jet will obviously affect the characteristics of atomisation and mixing of fluids. Data obtained from atomization experiments done on this jet using stimulant fluids (water in LOX side and air in GH_2 side) such as particle distribution and Sauter Mean Diameter (SMD), fairly matches with the predictions made using CFX software and associated codes which enabled to evaluate the analysis software. Further, the air flow at exit has been made axisymmetric by modifying the air passage in the injector jet (Jet 3 Mod) and the effect of this modification on atomisation and mixing of 2-phase fluids is predicted using the CFX software. In combustion better efficiency with less instability is to be attained. For meeting both simultaneously articles size distribution after injection has to be neither very finer nor coarse. Comparison of predicted data on atomisation revealed that the modified Jet (Jet 3 Mod) is relatively better to give stable combustion with higher performance.

Nomenclature

A = cross section area (m^2)
C_d = coefficient of discharge (s^{-1})
D = diameter or particles (microns)
Isp = specific impulse (s or Ns/kg)
L = length (m)
m = mass flow rate (kg/s)
P = pressure (bar)
ΔP = pressure drop (bar)
Re = Reynolds number

Greek Letters

μ = absolute or dynamic viscosity (Pa.s)/microns
φ = coefficient of contraction
ρ = density (kg/m^3)
σ = surface tension of drop (N/m)

Subscripts

a = air/axial
inj = injection
w = water
o = liquid oxygen (LOX)/outlet
f/h = gaseous hydrogen (GH_2)
R = total resultant
r = radial
θ = tangential/circumferential

Introduction

The design of the injector for 2-phase propellants (LOX/GH_2) injection in cryogenic rocket engine has been a problem of interest in the area of propulsion. Most of the injector configurations used in cryogenic engines are co-axial multi-element injectors with and without swirling of injected propellants with 4 different configurations (Jets 1 to 4) depending upon the total injector head assembly

* M.S. Research Scholar ** Assistant Professor
CEG, Anna University, Chennai - 600 025, India, Email : sujithmil@yahoo.com
Manuscript received on 22 Mar 2007; Paper reviewed, revised, re-revised and accepted on 29 Feb 2008

configuration and face plate cooling technique. Simulation of injection, mixing and atomisation processes in liquid rocket engines poses a variety of challenges. The atomisation of liquids by a gaseous stream is a phenomenon having fundamental physical importance. The objective of the injector jet design should be to ensure propellant mixing and atomisation so that the combustion efficiency will be high with stable combustion. The present study is on one type of injector jet (designated as Jet 3) for which some experimental data on atomisation of simulant fluids is available. Considering the criticalities in the development of cryogenic engine injectors, a design tool for injectors is essential to reduce development cost and effort, at the same time having higher reliability. Hence an efficient CFD code to analyze the mixing and atomisation of 2-phase propellants in front of injector face can improve the engine design and reduce the cost of development hardware. The objective of the present work is to develop an efficient CFD design tool for coaxial injector Jet 3 with partial swirl to the injected 2-phase fluids through correlation with experimental data.

Information on major operational cryogenic engine developed so far was obtained from literatures [1-5] along with their specifications and configurations which are used for the design of the proposed injector Jet 3. Atomisation and mixing studies done on similar injectors but without swirlers are discussed in [6, 7]. It includes both experimental and CFD studies on non-swirl type injector jets with fluid pairs like LN_2/GN_2 and water/air in place of LOX/GH_2. In the present study, the CFD analysis of atomisation and mixing of 2-phase propellants injected through designed injector jet was done in two stages using CFX software. (a) In the first stage flow analysis inside the injector jet to study the flow pattern at exit of jet which obviously affect the atomisation and mixing of 2 phase simulant fluids (water for LOX and air for GH_2) and comparison of predicted results with experimental data available for Jet 3 has been made. (b) In the second stage, the non-axisymmetric air flow at exit of Jet 3 has been made axisymmentric by modifying the flow passage in the injector jet (Jet 3 Mod) and the effect of this modification on atomisation on mixing of simulant fluids was predicted. Design of injector jets and their 3D-solid modeling and CFD atomisation analysis model for simulant fluids are presented. The predicted results have been validated with the results of experimental studies carried out at LPSC [8] on injector Jet 3.

Theoretical Studies

For smooth combustion of injected propellants in minimum chamber volume, the injector head should provide uniform distribution of propellants in the chamber cross section both in respect of velocity components and in terms of mass flux (ρV). The size of drops of liquid propellants injected to chamber should be as uniform as possible, so that the process of vaporization and mixing would be completed faster. In solving the above mentioned problems, the selection of injectors, their flow characteristics and the relative location of fuel and oxidizer injectors have great importance. Cryogenic engine using LOX/GH_2 as propellants uses injector head with multi-element coaxial injector elements (Jets). The injector elements are of coaxial type with LOX flowing centrally surrounded by GH_2 flowing through an annular passage. Usually testing and evaluation of a single injector element will be done before integrating them into a multi-element configuration for bigger engine testing. The specification, injector element configuration selection, detailed design, generation of fluid flow model and their CFD analysis are presented here.

Specification of injector element (Jet 3) used for the design.

Propellant combination (simulant fluids)	=	LOX/GH_2 (water/air)
Combustion chamber pressure (Pc) (bar)	=	60
Mixture ratio, (LOX/GH_2) by weight	=	6
Nominal GH_2 flow rate for combustion (g/s)	=	34.4 (with 60 bar Pc)
LOX flow rate (g/s)	=	224.4
GH_2 injector temperature, (K)	=	140
LOX injector temperature, (K)	=	95
GH_2 injector pressure (Nom), (bar)	=	70.2
LOX injector pressure (Nom), (bar)	=	69

Injector jet should be designed to feed propellants uniformly at intended condition into the combustion chamber, efficiently atomize and mix them to produce smooth and stable combustion with high performance. The following are the major considerations for designing a good injector [1-5].

- Injector element configuration
- Thrust per element/propellant flow rates
- Injector pressure drop
- Injection velocity ratio (V_f/V_o)
- Injection momentum ratio ($M_o V_o/M_f V_f$) = MR = $M_a V_a/M_w V_w$ (for water/air)
- Injector flow pattern
- Fabrication aspects

Jet 3 injector element shown in Fig. 1a (old design), Fig. 1b (modified) is provided with LOX swirl and non-swirl for GH_2. The inner sleeve has the main LOX stem having ID of 4.8 mm. Four tangential holes of ϕ1.7 mm drilled on the surface of inner sleeve at an offset distance of 1.45 mm with respect to injector axis imparts centrifugal motion to the fluid. The outer body is inserted to the inner sleeve and EB welded. The inner diameter of the outer body is 10 mm. Six numbers of straight grooves are cut on the inner sleeve for guiding the outer body into it. GH_2 enters from the top of the injector element through four numbers of EDM (Electro Discharge Machined) slots and flows out through the annular gap between inner sleeve and outer body without swirling. GH_2 orifice is designed for injecting 34.4 g/s of GH_2 at 70.2 bar and

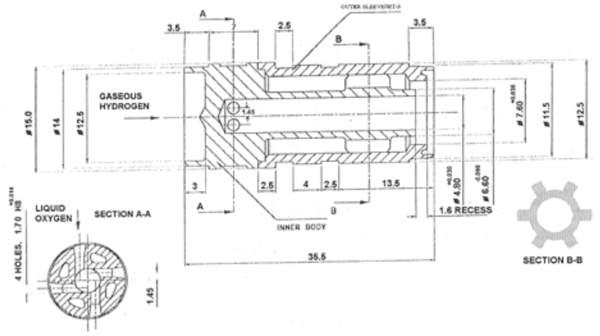

Fig.1a Injector assembly of Jet-3

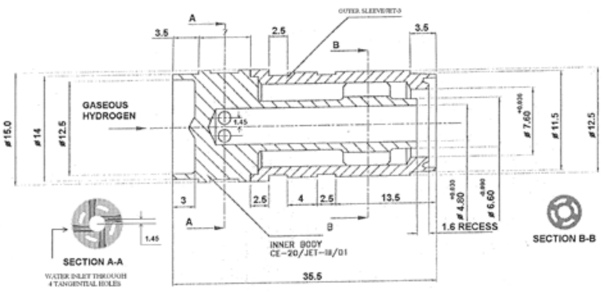

Fig.1b Injector assembly of Jet-3 Mod

140K into the combustion chamber without any swirl introduced in the flow passage of the jet so that the pressure drop should be 10.2 bar.

LOX element is designed using the engine hot firing conditions (Pc = 60 bar). 224 g/s of LOX at 69 bar and 95 K is injected into the combustion chamber with tangential swirl introduced in the flow passage of the jet so that the pressure drop should be 9 bar. The tangential swirl No.(A) and swirl parameters are derived using equations (1-3) taken from reference [5] and values are A = 1.204 and C_d = 0.4, Coefficient of flow cross section (Φ) = 0.6034, cone angle (α) = 83°.

$$A = \frac{R_{Bx} \times r_c}{n \times r_{Bx}^2} \tag{1}$$

$$C_d = -\frac{1}{\sqrt{\frac{A^2}{1-\phi} + \frac{1}{\phi^2}}} = \sqrt{\frac{\phi^3}{2-\phi}} \tag{2}$$

$$Tan\ \alpha = \frac{2\, C_d A}{(1 + \sqrt{(1-\phi^2)} - 4\, C_d^2 A^2)^{0.5}} \tag{3}$$

where R_{Bx} = distance of the axis of the entry holes from the axis of injector jet (1.45 mm), r_c = radius of jet at exit (2.4 m), r_{Bx} = radius of entry holes (0.85 mm), n = number of entry holes (4).

Hence momentum ratio of jet 3 at the normal operating condition is $M_h V_{ha}/M_o V_{oa}$ = 34.4 x 298.2/224 x 18.14 = 2.52

But the above momentum ratio is lesser than that of preignition momentum ratio of jet 3 which is around 3.08. Hence this value was taken for the experimental evaluation of atomisation as well as for CFD computational analysis using CFX (Table-2.).

Non dimensional parameter which has influence on atomisation and mixing of fluids is Weber number ($\rho h\, Vh^2\, Do/\sigma_0$). For simulation experiment and corresponding CFD analysis, it is 6.1 x 10^3 for jet 3.

Experimental Set-up for Atomization Study with Water and Air

Schematic layout of the setup made for atomization study with water and air is shown in Fig. 1c. It consists of

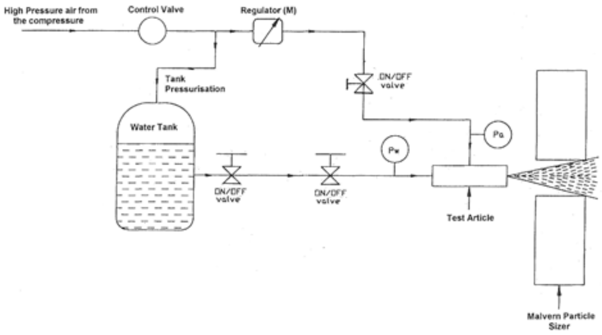

Fig.1c Schematic layout of experimental setup

pressure-fed systems for water and air and control valves and monitoring systems. The test article is suitably mounted so that the spray of water and air ejected from it can be visualized by Malvern laser light scattering equipment. The momentum ratio ($M_a V_a/M_w V_w$) is maintained same as the designed values of injector jets at pre-ignition condition.

Instrumentation and Measurement

Drop size distribution is measured using Malvern Laser Light Scattering instrument, which is the most sophisticated instruments available to measure, process and record the drop size data. The instrument is based on the principle of the **Fraunhofer Diffraction Theory** of a Collimated laser beam scattered (θ) by moving droplets (dia.D).

$$\theta \propto \frac{1}{D}$$

Detectors are semicircular plate with photosensitive band rings, which converts light energy to electric energy. The features are (a) each ring represents a range of drop size (b) intensity of light on any band ring is proportional to the number of drops in that range represented by (Vol %) (c) for drop size range from 1.2 μm to 1200 μm one detector can be used with lesser resolution. The scattered light intensities integrated over the finite detector areas digitized and processed in a computer to estimate drop size distribution. The series of 31 detectors in the detector array enable to receive signal and process it to yield the particle size. Use of different range lenses such as 1000 mm, 300 mm and 100 mm focal length helps the user to capture the signal from a wide range of particle sizes varying from 2000 μm to 0.5 μm from the same system of detector array.

Measurements in different cross sectional planes of the jets are carried out setting the Malvern instrument suitably. Then using the data obtained in different perpendicular planes of the jet axis, the particle distribution (Vol % vs drop size) at the central axial plane of jets are computed using the online computer.

Modeling of Injector Jets for Computational Analysis

The injector jet consists of two parts namely inner casing and outer sleeve. Each part is modeled in I-DEAS Software as shown in Fig. 2a and then they are assembled

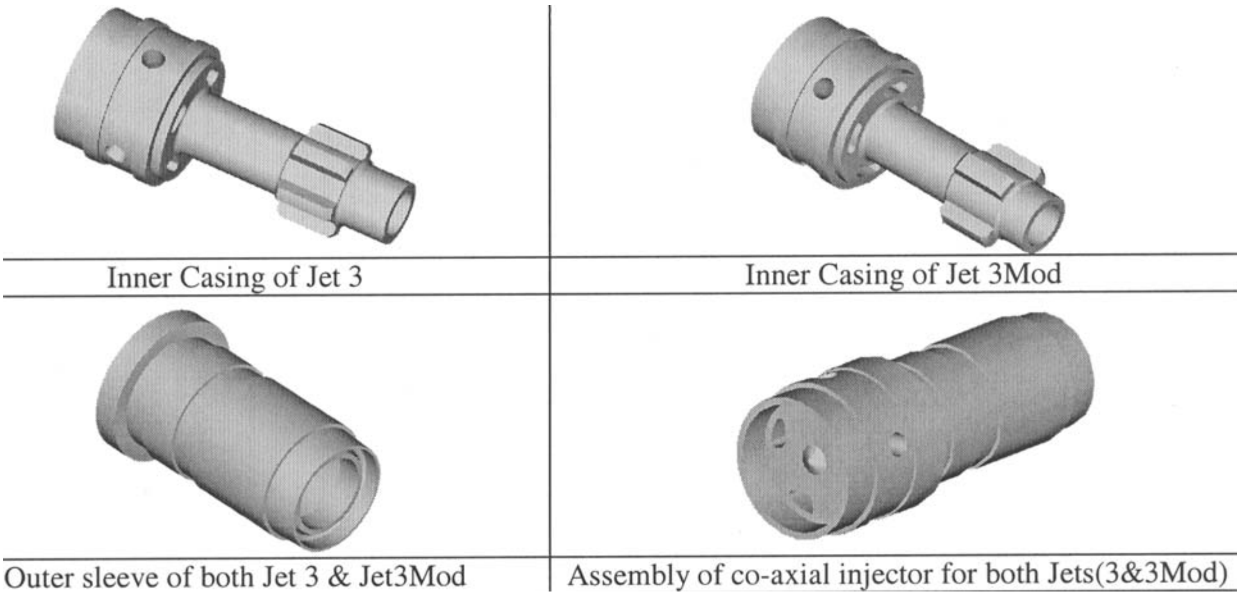

Fig.2a Injector jets-3 and 3 Mod modeled in I-DEAS software

as shown in the same figure bottom. This assembled model is the solid domain or the real injector named as Solid-1. From the above solid domain extract the outer surface only and make it into another solid part having no hollow space inside named as Solid-2. Through 'Boolean' operation subtract Solid-1 from Solid-2 to get the fluid domain as shown in Figs. (2b to 2d).

As the flow of the fluids inside the injector jets is not influencing each other, flow analysis can be done separately for each fluid. For that, separate the combined fluid domain into two fluid domains for the respective fluids (inner liquid and outer annular gas) by abstract operation in I-DEAS. Analysis are done for inner core fluid domain (liquid) for both jet 3 by giving suitable boundary conditions given in Table-2 and defining the physics for the domain to find out the output results (given in Table-2) as shown in Fig.2b Similarly annular fluid (air) domain analysis for jet 3 is done by giving suitable boundary conditions given in Table-1 to find out the output results shown in Fig.2c. These output results of both liquid and gas domains are used for the atomisation analysis.

The meshing details of injector fluid domain for both inner core fluid and annular fluid are given in Table-1.

Flow Path of Simulant Fluids in Co-axial Injector

As the water is swirling inside the inner core, the outlet velocity consists of three components-axial, radial and circumferential. The computational analysis is carried out for finding out these components, which can be used later for atomisation study. The boundary conditions given to

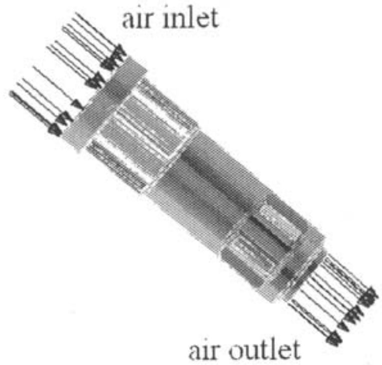

Fig.2c Annular fluid domain analysis for Jet-3

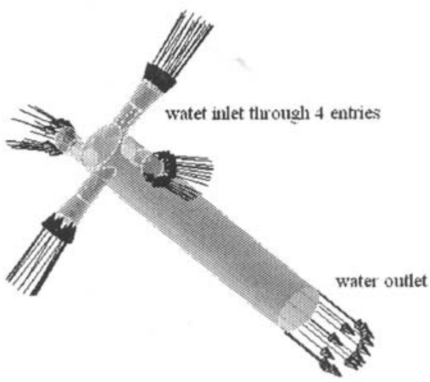

Fig.2b Inner core fluid domain analysis for both Jets (3 and 3 Mod)

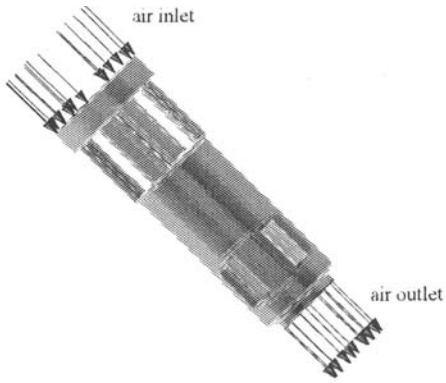

Fig.2d Annular fluid domain analysis for Jet-3 Mod

Table-1 : Meshing details for injector fluid domain analysis

	Inner Core Fluid Domain for Jet-3 and 3 Mod	Annular Fluid Domain	
		Jet-3	Jet 3 Mod
No. of Nodes	11444	59146	60977
No. of Elements	56694	297310	300427
No. of Tetrahedral	56694	297310	300427
Max. Edge length ratio	5.01056	7.00301	6.61643
Volume (m^3)	5.46296×10^{-7}	1.31061×10^{-6}	1.3158×10^{-6}

the fluids domain of injector for analysis are given in Table-2. The outlet velocities of computational analysis done on inner liquid domain with liquid domain with inlet parameters given in Table-2 are shown in Figs. 3a to 3h. Figs. 3a to 3d represent the resultant velocity distribution of liquid inside the jet. The average values of computed values of velocities at the outlet of the jet fairly match with the theoretical values given in Table-2. As the modified design of Jet-3 mod has no alteration in the inner core fluid, the output result is just as same as that of Jet-3. The output results of the computational analysis of water flow at exit of the jet shows axi-symmetric pattern as seen in Figs. 3e to 3h.

Computational Analysis of Outer Annular Space Fluid (Air)

While considering the air flow in the annular space of Jet 3, the results shown in Fig. 5a is nearly 180° symmetric as the air fluid domain (Fig.2c) itself is not symmetric because of the asymmetric design of the jet. This asymme-

Table-2 : Inlet and outlet parameters of water and air used as boundary conditions

Parameters and Fluids			Jet-3 and 3 Mod	
			Case 1	Case 2
Mass Flow Rate	Water, m_w (g/s)		67.7	54.5
	Air, m_a (g/s)		4.62	3.43
Inlet Velocity	Water, V_{iw} (m/s)		5.38	4.34
	Air, V_{ia} (m/s)		16.91	14.87
Outlet Velocity	Water (m/s)	V_{owa}	6.19	4.99
		V_{owr}	5.48	4.41
		$V_{ow\theta}$	4.346	3.51
		V_{owR}	9.34	7.53
	Air (m/s)	V_{oaa}	276.8	243
		V_{oar}	-	-
		$V_{oa\theta}$	-	-
		V_{oaR}	276.8	243
Inlet Pressure	Water, P_{iw} (bar)		1.925	1.60
	Air, P_{ia} (bar)		1.95	1.62

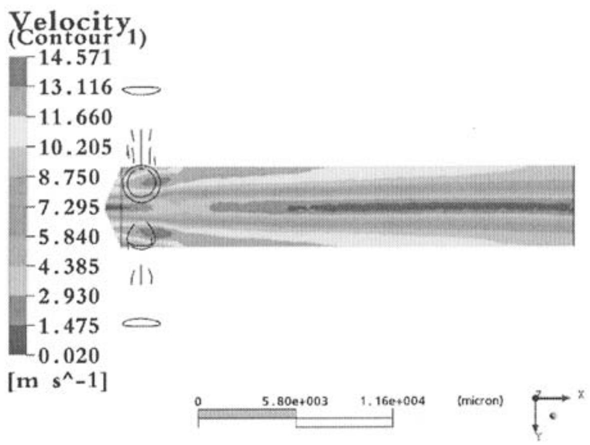

Fig.3a Water resultant velocity distribution at axial central plane

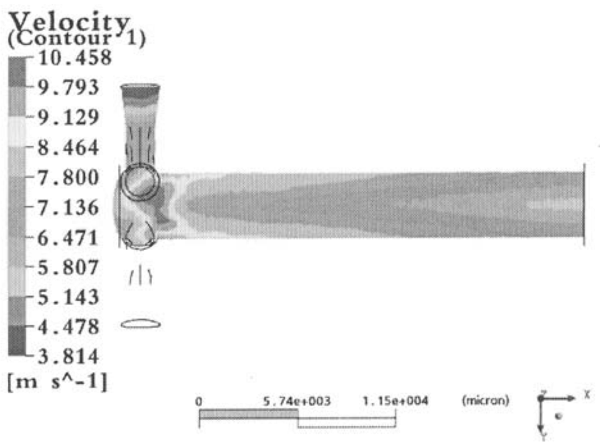

Fig.3b Water resultant velocity distribution at 1.45 mm off central plane

try is due to the flow passage. In the old design, the fluid is passing through six straight slots after 4 EDM holes which lead to the sudden blockage of fluid in certain places and free passage in other places.

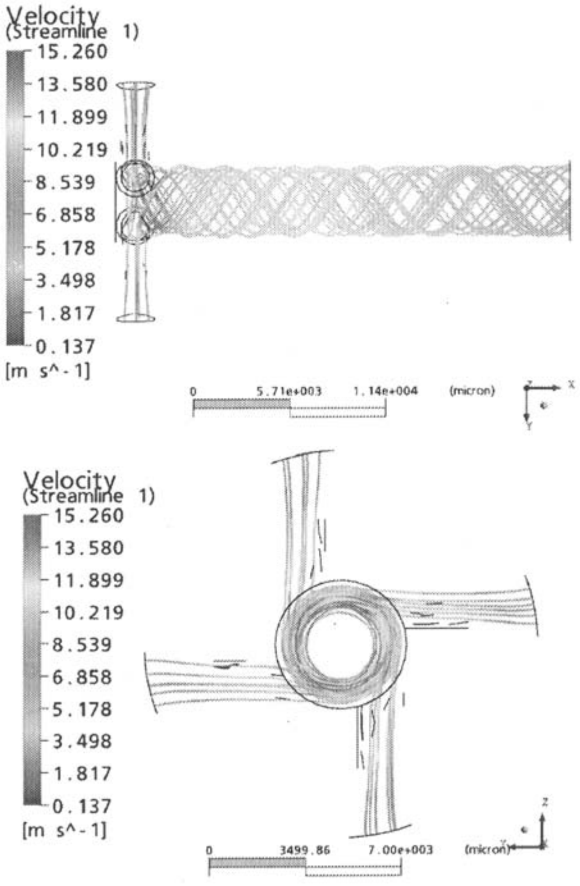

Fig.3c Water resultant velocity distribution

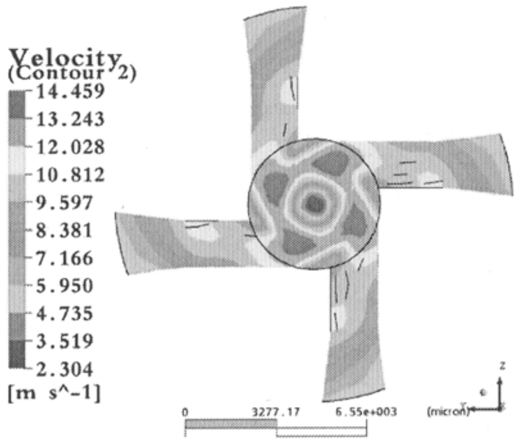

Fig.3d Water resultant velocity distribution at the swirling portion

At this instant, the original asymmetric configuration of the air fluid domain is modified to asymmetric flow configuration shown in Fig. 1b, without affecting the overall flow resistance for the specified mass flow rate. This modification is done by changing the 6 slots into 4 slots and making them inline with EDM entry holes and also the asymmetric shape of EDM holes is changed to symmetric shape keeping the same flow area. The results of the flow analysis carried out on the modified configuration are presented in Figs. 4b and 5b. Here similar to liquid (water) flow, an axi-symmetric flow field is obtained at the air exit of the modified jet unlike approximately 180° symmetric in the original jet 3 with 6 slots. This axi-symmetric flow of both fluids (water and air) will enable to achieve better atomisation and mixing of injected fluids which will be discussed below. For the modified jet (Jet 3 Mod) the relevant data made for analysis are shown in Figs. 2a, 2b, 2d, 3a to 3i, 4b and 5b.

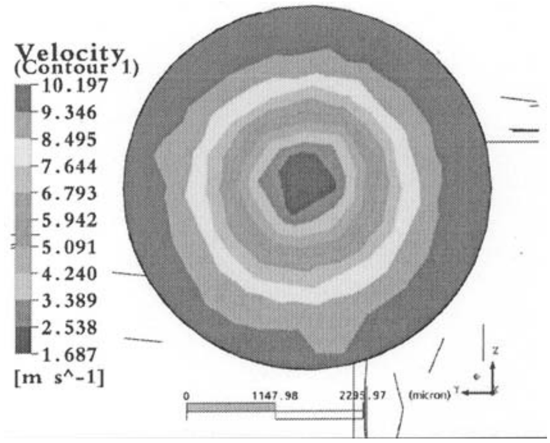

Fig.3e Water resultant velocity distribution at exit

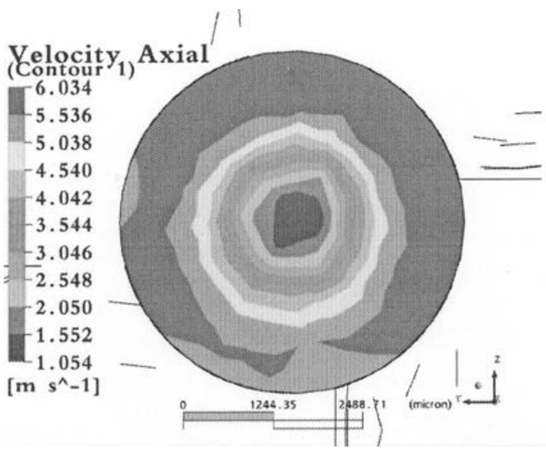

Fig.3f Water axial velocity distribution at exit

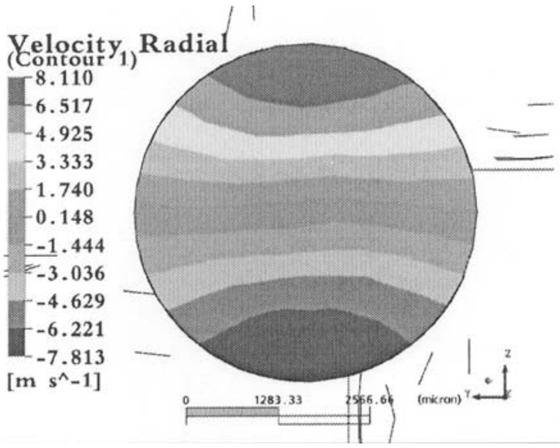

Fig.3g Water radial velocity distribution at exit

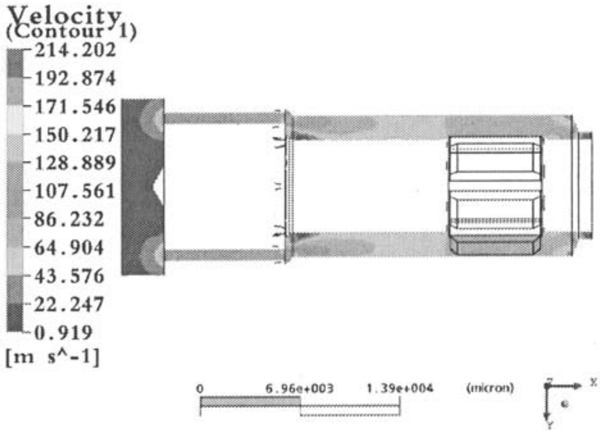

Fig.4b Air velocity distribution at axial plane inside of Jet-3 Mod

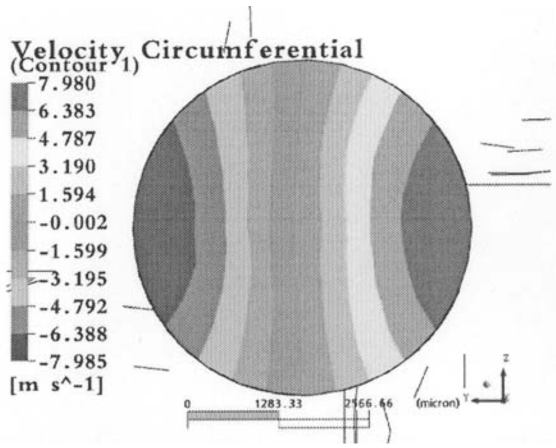

Fig.3h Water circumferential velocity distribution at exit

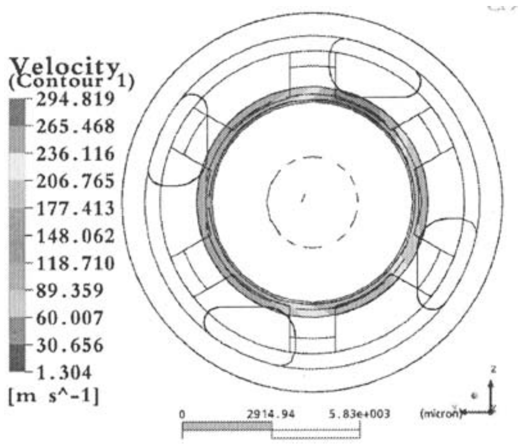

Fig.5a Air resultant velocity distribution at exit of Jet-3

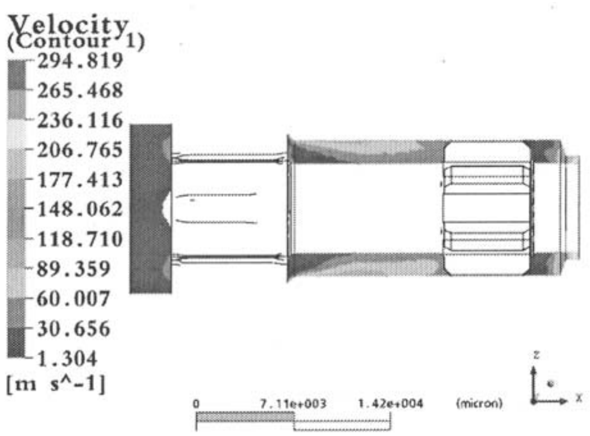

Fig.4a Air velocity distribution at axial plane inside of Jet-3

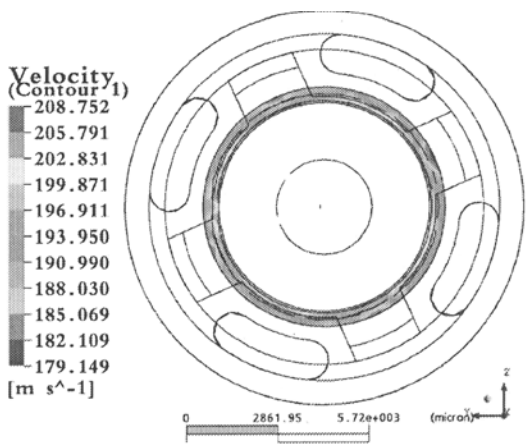

Fig.5b Air resultant velocity distribution at exit of Jet-3 Mod

Computational Atomisation Analysis with Water and Air Using CFX

The computational domain of open chamber for water and air jets is shown in Fig. 6a. In the case of water and air, both fluids are injected into the atmosphere. The meshing details are given in Table-3.

The meshed domain is given with suitable boundary conditions for both fluids at inlet to atmosphere. The atomisation analysis was done on the above atmospheric domain by using the output results obtained from the injector jet flow analysis.

From the atomisation analysis velocity distribution of the liquid particles were obtained. Using the velocity distribution the SMD can be found out by using Hiroyasu et al. formula [9, 10] given below. Both the formulae are used medium pressure injections. First is used for lower and second for higher injection velocities respectively.

$$Re_{jl} = \frac{\rho_l V_l d_j}{\mu_l}, \quad We_{jl} = \frac{\rho_l V_l^2 d_j}{\rho_l}$$

$$SMD = 4.12 \; Re_{jl}^{0.12} \; We_{jl}^{-0.75} \left(\frac{\mu_l}{\mu_a}\right)^{0.54} \left(\frac{\rho_l}{\rho_a}\right)^{0.18} d_j$$

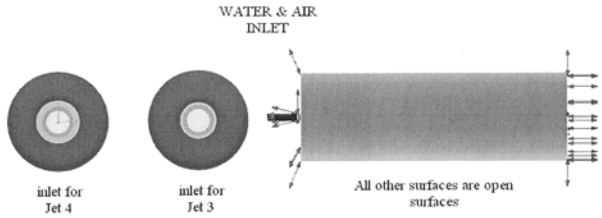

Fig6a Computational domain of open chamber for both Jets (3 and 3 Mod)

Table-3 : Meshing details for open chamber fluid domain analysis	
	Jet-3 and Jet-3 Mod
No. of Nodes	90314
No. of Elements	86400
Wedges	3600
Hexahedra	82800
Max. Edge Length Ratio	1443.82
Volume (m^3)	0.00235558

$$SMD = 0.38 \; Re_{jl}^{0.25} \; We_{jl}^{-0.32} \left(\frac{\mu_l}{\mu_a}\right)^{0.37} \left(\frac{\rho_l}{\rho_a}\right)^{0.47} d_j$$

where, Re_{jl} = Reynold's Number, We_{jl} = Weber Number, μ = Dynamic Viscosity, ρ = Density, d_j = Diameter of jet, σ = Surface tension

Subscript, 1 = Liquid, a = air

Drop size distribution can be estimated by Gaussian Normal distribution method given below :

$$f(d) = \frac{1}{\sigma\sqrt{2\pi}} \exp\left(\frac{-(d-\bar{d})^2}{2\sigma^2}\right) \quad -\infty < d < \infty$$

where, $\bar{d}$ is mean value and σ is the standard deviation

$$\bar{d} = \frac{d_{min} + d_{max}}{2} \qquad \sigma^2 = \frac{(d_{max} - d_{min})^2}{12}$$

d_{min} = Minimum diameter and d_{max} = maximum diameter of droplet at a particular velocity range region.

Using CFX software the velocity distribution of the atomized jet is only possible. Hence a code was developed to compute the SMD, drop size distribution and volume percent of drops from the velocity distribution obtained from the CFX atomisation analysis.

Results and Discussion

Results of Computational Studies done with Simulant Fluids (Water and Air)

Comparison of atomisation of Jet 3 and Jet 3 mod flow conditions are presented in Figs. 7a and 7b, which give the

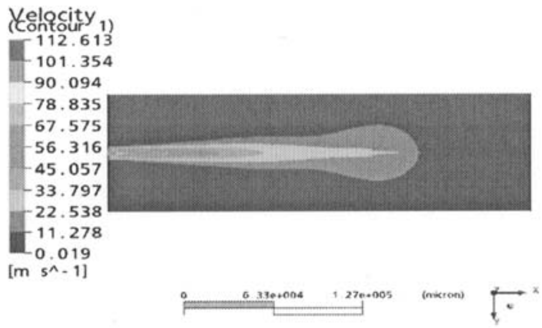

Fig.7a Computed velocity distribution of simulant fluid at central axial plant for Jet-3

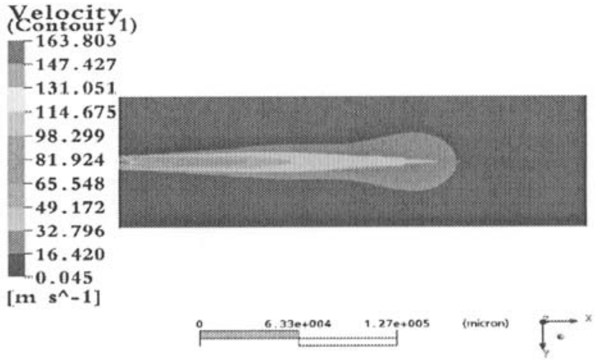

Fig.7b Computed velocity distribution of simulant fluid a central axial plant for Jet-3 Mod

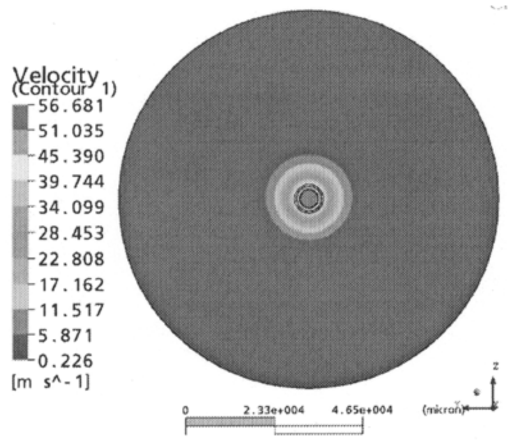

Fig.9a Velocity distribution of simulant fluid at 50 mm from inlet for Jet-3

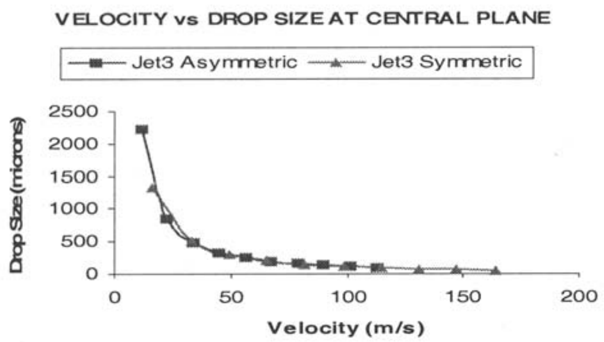

Fig.8a Variation of drop size with velocity at central axial plane for both Jets

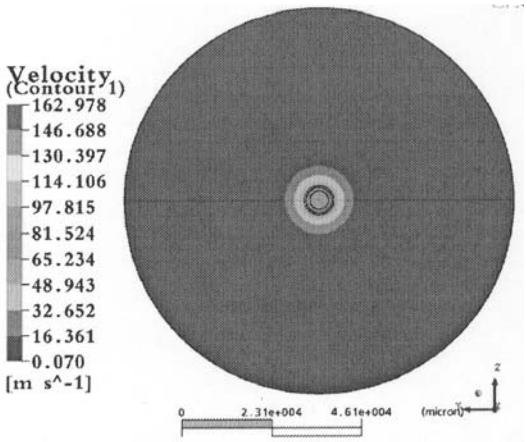

Fig.9b Velocity distribution of simulant fluid at 50 mm from inlet for Jet-3 Mod

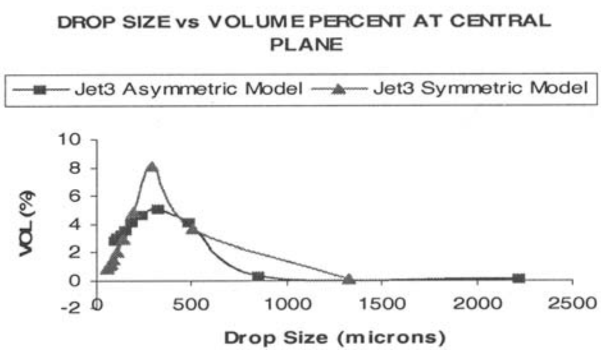

Fig.8b Drop size distribution at central axial plane for both Jets (3 and 3 Mod) for comparison of atomisation

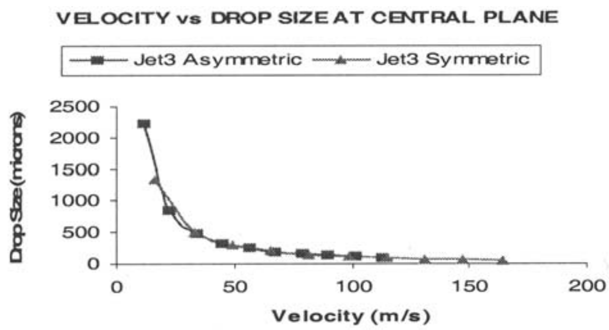

Fig.10a Variation of drop size with velocity at 50 mm from inlet for both Jets

velocity distribution at central axial plane of the spray. For the same case variation of drop size with velocity at central axial plane of the spray of both jets are given in Fig. 8a and droplet size distribution with their volume percent is presented in Fig. 8b. From this figure the requirements for better combustion efficiency and less instability can be met by modified jet, since it gives moderate size particles (300 μm) around 8% as compared to 5% in the original jet. Again the volume percentage of higher drop size from 600 to 1400 μm are relatively high in modified jet than in original jet which will improve the combustion stability. Similarly the velocity distribution at different planes perpendicular to the spray jet axis are also computed and the values at 50 mm from inlet are shown in Figs. 9a and 9 b for jets (3 and 3 Mod) respectively. These series of data reveal that the drop size become finer at longer distance and finally becomes zero. Further the variation of drop size (SMD) with velocity at different axial distances (cross sectional planes) are also computed and the values for plane 50 mm from Inlet of spray generated by Jets (3 and 3 Mod) are shown in Fig. l0a These data reveal that the particles become finer when the velocity of it increases. Similarly the drop size distribution with there volume percent computed at 50 mm cross sectional planes of both sprays produced by Jets (3 and 3 Mod) is shown in Fig. l0b. This data also confirms that Jet 3 Mod gives better combustion requirements as mentioned in abstract than Jet 3. Correspondingly the velocity distribution at cross section 100 mm from inlet of spray is shown in Figs. 11a and 11b and its drop size distribution with velocity and volume percentage is shown in Figs.12a and 12b respectively. From this data it reveals that even at 100 mm distance from inlet of jet more favourable particle distribution can be obtained by Jet 3 Mod than Jet 3.

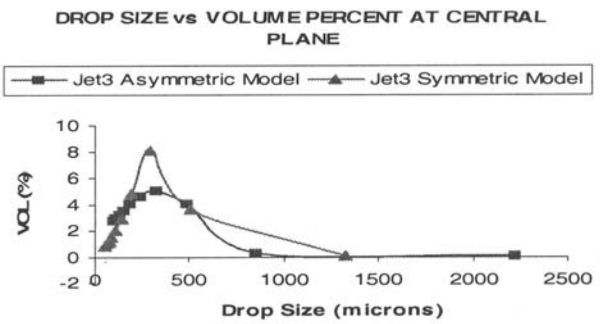

Fig.10b Drop size distribution at 50 mm from inlet for both Jets

Correlation of Analysis with Experimental Data and Evaluation of Merit of Jet 3Mod

The computational results of Jet 3 are compared with its experimental data and are shown in Figs. 13a and 13b. In the experiment it was possible to measure drop sizes right from 1 μm to 1120 μm in the spray produced by Jet 3 where as in the computational analysis, drop sizes below 90 μm could not be obtained as seen from Fig.13a. However in the computational range, reasonable matching of the data (within 20%) is found. From Fig. 13b it seen that it was possible to measure drop sizes from 1 μm to 920 μm in the spray of Jet 3 at 50 mm cross sectional plane, where as in computational analysis drop sizes below 220 μm could not be obtained. However in the computed

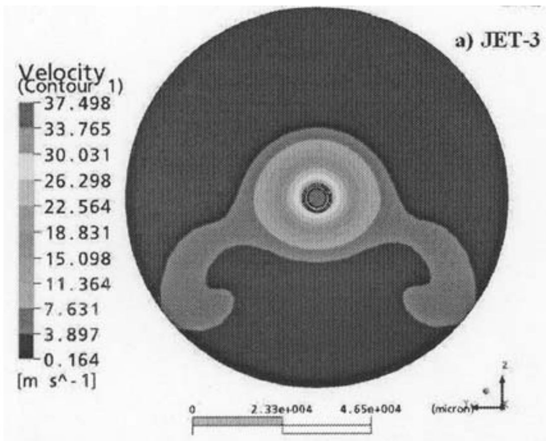

Fig.11a Velocity distribution of simulant fluid at 100 mm from inlet for Jet-3

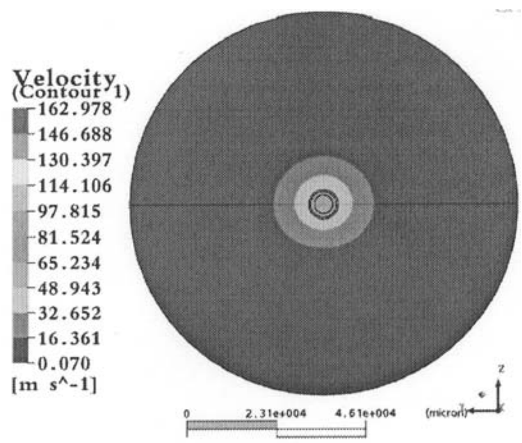

Fig.11b Velocity distribution of simulant fluid at 100 mm from inlet for Jet-3 Mod

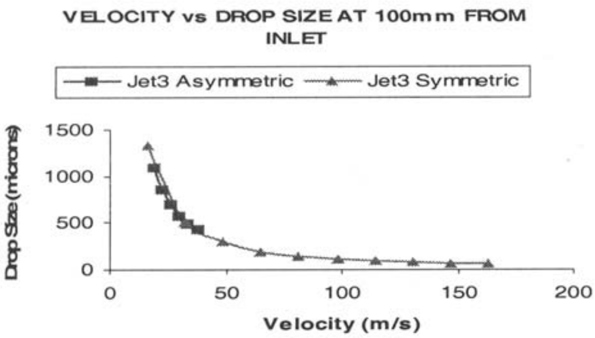

Fig.12a Variation of drop size with velocity at 100 mm from inlet for both Jets

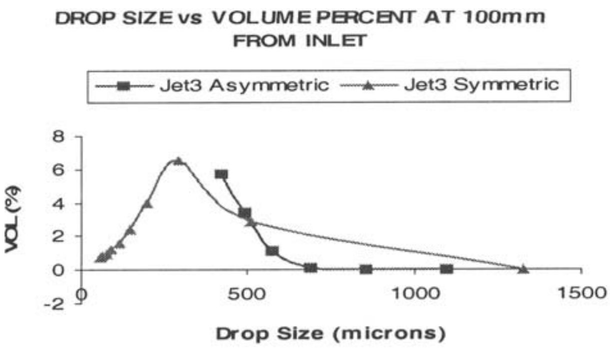

Fig.12b Drop size distribution at 100 mm from inlet for both Jets

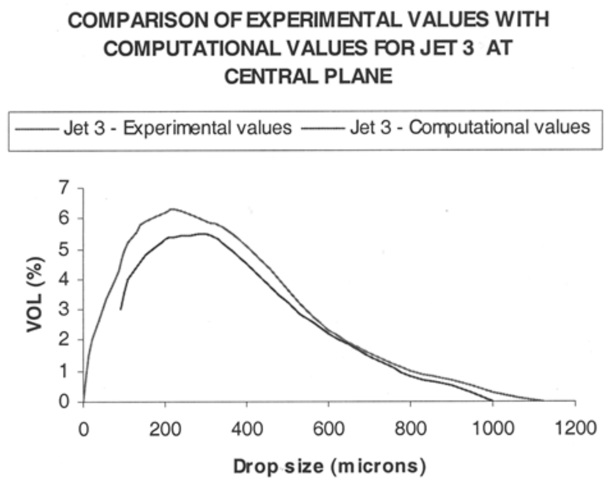

Fig.13a Comparison of computational data with experimental data at central axial plane of Jet-3

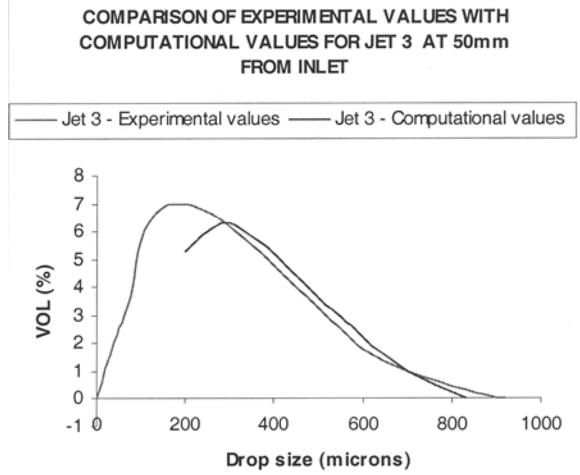

Fig.13b Comparison of computational data with experimental data at 50 mm cross section of spray of Jet-3 and Jet-3 Mod

range, here also reasonable matching of the data (within 20%) is found.

Hence the software used for the prediction of atomization pattern of jet 3 can be considered as evaluated. The atomization analysis of the spray of Jet 3Mod is also done with the same software and hence their relative merit in atomization of 2-phase fluids can be judged from the results presented in Figs. 8b and l0b.

Conclusion

The results of atomisation studies carried out on a cryogenic injector jet, which are designed and evaluated using simulated fluids (water and air), are used for the purpose of evaluation of software. Suitable computer codes are developed for the post processing of the velocity distribution computed, like estimation of drop sizes (SMD) and then distribution of drop sizes in the spray along with its presence in the form of volume percent. Flow analysis of both fluids through the injector using CFD revealed axi-symmetric flow at exit of the LOX (water) passage whereas asymmetric flow at the exit of GH_2 (air) passage. Further modification done on the annular air passage of the injector jet (Jet 3Mod) and CFD analysis on the modified air fluid domain gave axisymmetric flow at the exit of GH_2 passage also. Due to the axi-symmetric flow through both passages in Jet 3Mod a more favourable and uniform droplet distribution is achieved from combustion point of view.

References

1. Liquid Propellant Rocket Engines, NASA-SP 125.
2. Sutton, G.P., "Rocket Propulsion Elements", John Wiley and Sons, Inc, New York, 1956.
3. "Liquid Propellant Rocket Injectors", NASA - SP-8089.
4. David T. Harrje., "Liquid Propellant Rocket Combustion Instability", NASA-SP-194.
5. Dobrovolsky, M.V., "Liquid Fuel Rocket Engines-Design Fundamentals", FTD-MT.24-I35-69-Vol.1 and Vol. 2.
6. Ranger, A. A. and Nicholls, J.A., "Atomisation of Liquid Droplets in a Convective Gas Stream", Int. J. Heat Mass Transfer, Vol. 15, 1972, pp.1203-1211.
7. Tong, A.Y. and Sirignano,W.A., "Analytical Solution for Diffusion and Circulation in a Vaporizing Droplet", Nineteenth Symposium (International) on Combustion/The Combustion Institute, 1982, pp.1007-1020.
8. Jayan, N. and Babu, R, "Report on Preliminary Atomisation Study Done on CE-20 Thrust Chamber Injector Elements", Report No. LPSC/C 25/TR/007/02.
9. Gray L. Borman and Kenneth W. Ragland., "Combustion Engineering", McGraw-Hill International Editions, Mechanical Engineering Series.
10. Hiroyasu, H., Arai, M. and Tabati, M., "Emperical Equations for the Sauter Mean Diameter of a Diesel Spray", SAE Paper No. 890464, 1989.

Nishant in Flight

Nishant is ready. At the Aeronautical Development Establishment (ADE) in Bangalore, four of the Unmanned Aerial Vehicles (UAVs), plus a Hydro Pneumatic launcher and a ground control station are all getting the final touches before they are tested and handed over to the Indian Army in about a month. Part of a Rs. 235 Crore order for 12 of these tactical UAVs that the Army had placed, these limited series production aircraft have been developed in-house jointly by ADE, National Aerospace Laboratories (NAL) and Vehicle Research and Development Establishment (VRDE), Ahmednagar.