Flow Field Analysis of a Liquid Ejector Pump using Embedded LES Methodology

Qasim Zaheer
qasim1985@cae.nust.edu.pk
Department of Aerospace Engineering, CAE, NUST, Risalpur 24080, Pakistan

Jehanzeb Masud
jehanzeb.masud@mail.au.edu.pk
Department of Mechanical and Aerospace Engineering, IAA, Air University, Islamabad 44000, Pakistan

ABSTRACT

The understanding of entrainment and mixing phenomenon in the ejector pump is of pivotal importance for designing and performance estimation. In this paper, the existence of turbulent vortical structures due to Kelvin-Helmholtz instability at the free surface between the motive and the entrained fluids streams are simulated using Embedded LES methodology. The efficacy of Embedded LES for simulation of complex flow field of ejector pump is evaluated using ANSYS Fluent®. The enhanced mixing and entrainment process due to breaking down of larger eddies into smaller ones as a consequence of Vortex Stretching phenomenon, is captured in this study. Moreover, the flow field characteristics of ejector pump like pressure velocity fields and mass flow rates are analyzed and validated against the experimental results.

INTRODUCTION

The Ejector Pumps are devices used to increase the pressure of bulk of the fluid by introducing or injecting a high speed motive flow. The increase is pressure is obtained by the momentum transfer between the motive fluid and the entrained fluid. Due to the simplicity of geometrical construction, ease of usage, less maintenance requirement and ease of installation has led to various practical applications of such pumps in industrial as well as engineering field. Though, these pumps are less efficient than their counterparts like centrifugal and displacement pumps however they carry the ability to handle large volumes of fluid. In the present study, the modeled ejector pump is used in transferring fuel between the fuel tanks of fighter aircraft.

The flow field of ejectors is quite complex as it is composed of free shear layers, jet flow and turbulent mixing region, the simplified analytical methodologies have limited capability for accurate performance prediction of these pumps. The experimental analysis of these pumps is not an economically feasible solution. However, with the advent of computational fluid dynamics, more precise results of complex flow field inside ejector pumps are possible by using turbulence modeling techniques in CFD simulation. But, each turbulence model has its own limitations and not every model can be used for all types of turbulent flows.

The literature survey indicates various studies conducted to analyze the flow field of ejector pump using different turbulence models. An insight into the ejector flow phenomena was obtained using computational and analytical tools and the results were compared via shadowgraph images of flow domain [1]. Computationally, the flow field was simulated using k-ε RNG and k-ω SST models, and the results revealed that later turbulence model predicted the flow features more accurately. The performance characterization of "short" ejectors was conducted analytically and experimentally and it was concluded that proposed new ejector model for "short" ejectors can accurately predict its performance [2]. Experimentally, Laser tomography visualization technique was also used to understand the air flow inside a supersonic ejector [3]. The focus was given to the choking of the flow and the results were used to validate the CFD results. The visualization of flow field of ejector pump (when integrated with Pulsed Detonation Engine for self-aspiration) using experimental techniques helped in ascertaining the fill fraction and equivalence ratio which successfully induce the secondary flow [4]. The effect of different geometrical configurations of primary flow inlets on the turbulent flow regime of jets was investigated using Reynolds Stress Model and it was concluded that development of the triangular jets is stronger than others [5].

The numerical simulation of complex flow phenomena, like ejector pump flow field, using RANS turbulence modeling techniques lacks the capability to identify the important flow features [6]. The accurate simulation of dynamics and physics of turbulent eddies is necessary to understand the flow characteristics of ejector pump as these turbulent eddies are responsible for mixing and entrainment mechanism. These important flow features cannot be simulated using RANS turbulence modeling technique as only time averaged flow field is analyzed whereas the turbulent vortical structures are time transient in nature. Apart from visualization of eddies, the correct estimation of turbulent viscosity associated with these turbulent structures is also important for accurate simulation of pressure and velocity fields of ejector pump. Again, the RANS models show their inadequacy for numerically simulating the flow characteristics when compared with the experimental results [7]. This is because the model coefficients used in mathematics of the RANS models are not calibrated with respect to complex flows like shear layers, jet flows and mixing regions which are the hallmark of ejector pump flow field. Hence, there arises a need to recalibrate the model coefficients to keep the results in good agreement with the experimental values [8] .

In literature, Large Eddy Simulation (LES) technique was used to numerically replicate the flow field of ejector pump as in LES, the flow parameters are spatially averaged instead of time averaged [9]. The results obtained were in good agreement with the experimental one. However, the fine mesh resolution requirement for performing good quality LES presents two major problems. Firstly, if LES is performed in the complete flow domain of ejector pump, then it is no more a computationally viable solution as the computational cost increases manifold. Secondly, if the associated computational cost is to be reduced then the domain of ejector pump has to be reduced. To obtain the boundary conditions at these reduced domain inlets and outlets is again a tedious job. Therefore, to overcome these limitations of LES, the Embedded Large Eddy Simulation (Embedded LES), a hybrid RANS-LES technique, is used in this study to simulate the ejector pump flow field. The region of interest where the turbulence is more pronounced, is resolved using LES whereas, the remaining flow is simulated or modeled using Standard k–ε RANS turbulence model.

TURBULENCE MODELING

Computational Fluid Dynamics simulations use conventional RANS models to simulate and most of the industrial turbulent flows. However, for complex flows, like ejector pump flow field, the performance of RANS models is much weaker [10]. With the passage of advancement in numerical techniques and computational power, it is now possible to simulate full turbulence spectrum in numerical domain. In LES, the large eddies which carry most of turbulent kinetic energy of the flow are resolved directly by solving the Navier Stokes Equation and the smaller eddies at which the dissipation occurs are modeled as they are homogeneous and isotropic. In case of free shear flows, as is the case in ejector pumps, it is normally much easier to resolve the largest eddy scales, as they are of scales of the thickness of the shear layer. As the flow field of ejector pump comes under the genre of locally unstable flow [10], hence in this study, Embedded LES technique is used for simulation. In hybrid RANS / LES methods, the problem of high computational cost is alleviated by blending the Large Eddy Simulation with RANS Models. ELES is therefore not a new turbulence model but in fact it is a combination of RANS and LES models joined by appropriate interface conditions. In ANSYS Fluent®, an Embedded LES formulation is available [11].

In this study, the Standard K-ε turbulence model is used in the RANS zone where as WALE subgrid scale in LES zone of ejector pump. Mathematically, the Standard K-ε is a semi-empirical model based on transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The transport equation is given by

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \overline{u_i}\ k)}{\partial x_i} = -\rho\ \overline{u'_i u'_j}\frac{\partial \overline{u_i}}{\partial x_i} - \rho\epsilon + \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right] \qquad (1)$$

and the dissipation rate equation is modeled as

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho \overline{u_i}\ \epsilon)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1\rho_k\frac{\varepsilon}{k} - C_2\rho\frac{\varepsilon^2}{k} \qquad (2)$$

the dissipation rate is related to TKE (k) and a turbulence length scale L_t:

$$\varepsilon \sim \frac{k^{3/2}}{L_t} \qquad (3)$$

The eddy viscosity μ_t can be expressed in terms of TKE (k) as:

$$\mu_t = \rho C_\mu L_t\sqrt{k} = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (4)$$

The constants carry the default values : C_μ=0.09, C_1=1.44, C_2=1.92, σ_k=1.0 and σ_ε=1.3.

The LES utilizes the implication of Kolmogorov theory that is the large eddies of the flow are dependent of geometry of the problem whereas the smaller ones are more universal in nature. This feature permits to explicitly solve the large eddies in calculation and at the same time, implicitly account for the small eddies by using a sub grid-scale model. Mathematically, the decomposition of resolved and modeled velocity field is attained by incorporation of filtering function G such that the velocity field decomposes into a resolved scale part and subgrid scale modeled part. The function is given by:

$$\overline{u_i}\ (\vec{x}) = \int G(\vec{x} - \vec{\epsilon})u(\vec{\epsilon})d(\vec{\epsilon}) \qquad (5)$$

This result in

$$u_i = \overline{u_i} + \acute{u_i} \qquad (6)$$

Here $\overline{u_i}$ is the resolved scale part of velocity vector where as $\acute{u_i}$ is the subgrid scale modeled part. The LES generally apply the grid cell dimension as the filtering operator (box filter). Hence the maximum cell dimension determines the length scale up to which the turbulent flow features are resolved and below this limit the modeling takes place. The incorporation of this decomposition in Navier Stokes equations results in nonlinear advection terms.

$$\tau_{ij} = \bar{u_i}\bar{u_j} - \overline{u_i u_j} \qquad (7)$$

The WALE subgrid scale model applies the Boussinesq hypothesis and seek to calculate (the deviator part of) the SGS stress.

CONFIGURATION AND COMPUTATIONAL SETUP

In the present study, the modeled ejector pump is used in aircraft fuel system for fuel transfer operation between the tanks. It immersed inside the fuel stored in the fuel tank. This fuel enters the ejector pump vide secondary fluid inlet at the hydrostatic pressure. The high pressure primary (motive) fluid from engine mounted Active Fuel Pump enters through the primary fuel inlet. The high-velocity fluid takes along any other fluid it passes through. High pressure stream of active fuel enters the pump from inlet. Active fuel stream carries the fuel from the fuel tank with it directly into the outlet pipe at a higher pressure and, at the same time, creates a vacuum that draws more fuel into the pump from the fuel tank. The different parts of the subject ejector pump are annotated in the Figure 1.

The primary nozzle inlet diameter is 18mm whereas the exit diameter is 7mm. The inlet diameter of the S-section is 50mm and the mixing chamber is having the constant diameter of 34 mm. The length of mixing and diffuser sections is 272mm and 271 mm respectively. The flow exits the pump having the diameter of 72mm. The geometrical dimensions of the ejector pump are translated into the 3D CAD Model of subject pump. Gambit® software is used for CAD modeling as well as meshing purpose. The complete flow domain of the ejector pump is divided into RANS zone and LES zone. The region of ejector flow between the primary nozzle exit and the mixing chamber exit is solved numerically by LES technique whereas rest of the domain by Standard K-ε turbulence model. The RANS-LES and LES-RANS interfaces are placed cautiously so that they must not be located in the region of interest i.e. they are located in the region of undisturbed equilibrium. The RANS-LES interface is placed at the inlet of S-section i.e. the cross section of ejector pump where the primary nozzle outlet is situated and the LES-RANS interface is placed at the exit plane of mixing section. To convert the modeled turbulent kinetic energy in the RANS zone to resolved one for LES zone, synthetic turbulence is generated at RANS-LES interface by using Vortex Method [12]. However, at LES-RANS interface no perturbation is generated synthetically.

The maximum cell dimension of numerical grid is approximated by analyzing the turbulent kinetic energy (k) and dissipation rate (ε) distribution using a precursor RANS simulation and then calculating the integral length scale over the LES zone of ejector pump. The integral length scale is selected as a reference because if the size of turbulent eddies of the order of integral length scale is resolved then almost 80% of the total TKE of the flow regime is considered to be resolved by LES. The integral length scale (l_o) is given by the equation (8) and the distribution of this scale on the grid using a precursor RANS simulation is shown in Figure 3.

$$l_o \sim \frac{k^{3/2}}{\varepsilon} \qquad (8)$$

The distribution of integral length scale helps in ascertaining the maximum cell dimension of grid used in LES zone. Therefore, a maximum cell dimension of $\mathbf{5 \times 10^{-4}}$ m is selected. The structured mesh is generated in complete domain of the ejector pump using hexahedral elements. A total of 2.44 million mesh cells in RANS zone and 9.8 million mesh cells in LES zone are generated. The details of the mesh structure can be ascertained from Figure 4. Keeping the CFL~1 requirement for LES, the time step of 5µsec is used with a total number of 20 sub iterations for initial time steps. Later on, as the solution progressed, the number of sub iterations for each time step is reduced. The fuel used as the working fluid of ejector pump is considered incompressible and the physical properties of jet A-1 fuel are used for simulation purpose. The pressure inlet boundary conditions are used for primary and secondary nozzle inlets and the gauge pressures of 0.6 MPa and 0.00224 MPa are defined at primary and secondary nozzle inlets respectively. The pressure of 0.025 MPa is defined as pressure outlet of the ejector pump. These values are extracted from the experimental data of the subject pump **[8]**. The instantaneous flow velocity at mixing section inlet and outlet is monitored to ascertain the statistically steady state of turbulent flow. Once the flow becomes statistically steady, the unsteady statistics are then gathered by simulating the flow field for another few flow through times.

RESULTS AND DISCUSSION

Grid Resolution Assessment

The quality of the Large Eddy Simulation embedded inside a global RANS domain is of utmost importance as far as the validation of results is concerned. It is believed that in general the LES results are in better agreement with the experimental evidences compared to that of RANS, especially when the flow field under investigation is complex. However, the accuracy is based on the fineness of the grid employed. But at the same time the grid independence methodology cannot be employed for the case of LES as the grid independent LES is actually DNS. Celik et al suggests eddy viscosity ratio as the indicator for assessment of grid resolution incorporated in the LES zone of ejector pump [13]. The LES Index of Quality (LES IQ) relates the turbulent viscosity to that of laminar viscosity using the expression given in equation (9).

$$LES\ IQ_{\vartheta} = \frac{1}{1+0.05\left[\frac{(\mu+\mu_{sgs})}{\mu}\right]^{0.53}} \qquad (9)$$

The LES IQ is a dimensionless number and it varies from 0-1. The constants of this expression are calibrated such that the index behaves similar to the ratio of resolved to total turbulent kinetic energy. The LES IQ greater than 0.8 is considered a good LES whereas 0.95 and higher is considered as DNS [13]. The LES quality index distribution throughout the LES zone of ejector pump is shown in Figure 5. It is evident that the grid resolution used satisfies the LES quality of index criteria. Hence, the simulated flow field using Embedded LES technique is regarded as sufficiently resolved flow regime.

Qualitative Analysis of Ejector Pump flow field

The flow pattern inside the ejector pump is categorized as complex flow in various literature as various fluid flow phenomenon occur at the same time. The flow field of subject pump is characterized by existence of adverse pressure gradient as the bulk of the fluid (secondary or entrained fluid) flows in the direction of increasing total pressure. Then there exists turbulent shear flows like mixing layers, turbulent jet flow phenomena and free shear layers. When a jet of viscous fluid enters at low velocity into the fluid at rest, a regular succession of vortices is given off at the rim or the exit of nozzle. As the velocity is increased beyond the certain limit, the periodic flow is replaced by turbulent mixing region. This irregular fluctuating flow region gives rise to momentum transport between the neighboring fluid regions and as a result viscous stresses will generate between these regions. This region is called shear layer / mixing layer of fluids.

According to Kelvin Helmholtz theory, there exists instabilities in the shear layer / mixing layer [14]. As the shear layer is inherently unstable, as described and proved by Kelvin Helmholtz, some disturbance will grow and form vortical structures. These structures develop as the shear layer "rolls up" due to the difference in the velocity magnitudes of primary and secondary fluid. These vortical structures merge into each other and develop large coherent structures which travels into each stream. Thus the fluid of one stream gets transported far out into other stream. This phenomenon is called "convective transport". The momentum transfer / energy transfer between two streams of fluid by this process is very much different from that of much slower process of molecular diffusion across the boundaries of neighboring fluid stream.

In order to identify the important characteristics of ejector pump flow field like formation of turbulent coherent structures, jet flow region and the entrainment phenomena, the "Q-criterion" vortex identification technique is applied to the LES zone of subject flow field. The "Q-criterion" is given by the expression (10) and is actually the second invariant of the velocity gradient tensor. In compressed form, Q parameter is given by:

$$Q = \frac{1}{2}(\Omega^2 - S^2) \qquad (10)$$

Once the "Q" parameter is computed in the whole domain, it consists of both the negative as well as positive values. The negative values of Q refer more strain rates than vorticity rates and vice versa. So by taking the positive values of Q, one identifies the flow region with more vorticity than the strain which is the inherent feature of turbulent coherent structures. This is implemented in the current study by creating the iso-surface of Q-criterion of having values around 5% of the maximum positive value of Q in the whole domain [15]. The iso-surface of Q-criterion is colored by the instantaneous velocity magnitude for better visualization of turbulent structures.

Focusing on the turbulent structures visualized in the Figure 6, it can be easily identified that certain class of coherent structures exists. One of such is the "Ring Vortices" which are generated in the near field of primary nozzle exit. At the nozzle exit, the annular shear layer is formed between the primary and secondary fluid. Because of the presence of cross directional Kelvin Helmholtz instability, the shear layer rolls up immediately downstream. The appearance of the vortices is linked to the start of turbulent kinetic energy increment. These vortex rings are convected downstream and they undergo pairing or merger procedure before breaking up into smaller vortices as depicted in the iso surface of Q criterion.

Another class of turbulent structure which is also visible in Figure 6 is the formation of V shaped vortices. These vortices are generated due to azimuthal instability of the vortex rings. Such vortices are responsible for the enhanced mixing between the two interacting fluid streams.

The presence of such turbulent vortical structures immediately downstream of the primary nozzle exit confirms the good mixing process representation at the edges of the primary nozzle jet. The breaking up large vortices into smaller ones suggests the mechanism of creation of turbulence which leads to enhanced mixing and entrainment process. The capturing of these vortical structures in not possible using RANS turbulence models. Moreover, the unsteady turbulent structures are present / generated at the expected locations which also supports the correct placement of RANS-LES interface.

Flow field Analysis of Ejector Pump

In order to validate the numerical scheme used for flow field simulation of ejector pump, the results of experimental setup of same ejector pump used by J. Masud and Javed (2007) [8] are taken as reference. The flow field is simulated for three test cases as depicted in the Table 1. The error estimation between the experimental and numerically simulated results signifies the validity of numerical scheme.

Pressure Field Analysis

The instantaneous static pressure field of ejector pump is visualized as contour plot in Figure 7 supported by the mean static pressure profiles along the centerline of the ejector pump in Figure 8. The mean static pressure along the centerline 'Px' is normalized by secondary nozzle inlet pressure 'Ps'. The time averaged mean values of static pressure is obtained after gathering the unsteady statistics of the flow domain. There exists a pulsating variation of static pressure in the stream wise direction of ejector pump. This pulsating pattern is due to the formation of turbulent coherent structures. By comparing the contour plots of static pressure variation and the iso surface of Q-criterion, it is evident that where the flow is turbulent, the static pressure deficit is an indicator of the phenomenon of turbulence [16]. The region of low pressure may be linked to generation of turbulence and to the effect of turbulence on the transport of matter and energy [17].

Immediately downstream of the primary nozzle exit, there is a sharp decrease in the static pressure of the motive flow along the centerline. The pressure remains negative throughout the turbulent region of ejector pump flow field. The recovery of static pressure from its negative peak is due to the mixing and momentum transfer processes between the motive and the entrained fluid. In the diffuser section of ejector pump, the static pressure of the flow is then increased to the design outlet pressure.

Velocity Field Analysis

The instantaneous velocity contour of the ejector pump, Figure 9, depicts a typical distribution of velocity field. The mean velocity contour plot and its variation along the centerline is depicted in Figures 10 & 11 respectively. The mean velocity parameter 'Ux' is non-dimensioned by primary nozzle exit velocity 'Un'. The high speed motive flow ejects from the primary nozzle exit and enters the relatively static secondary flow domain. While moving downstream, the motive flow jet transfers its momentum to the secondary flow and hence its velocity magnitude reduces. Moreover, due to the inherent instability of shear layer, the generation of turbulent structures entrains the secondary fluid into the primary fluid and enhances the mixing of primary and secondary flow streams. As a result, the shear layer grows downstream, consequently the jet spreads laterally outwards and the velocity of primary flow decreases along the centerline of the ejector pump. A few distance down the primary nozzle exit where the magnitude of velocity almost remains uniform is the "potential core". Due to the spreading of shear layers, the potential core eventually vanishes. The mixing process of the flow continues even in the mixing section of ejector pump and same is also evident from the velocity contours. The decay rate of the centerline velocity also characterizes the mixing level of the jet.

Turbulent Kinetic Energy Distribution Analysis

The essential feature of ejector pump like mixing and entrainment process between the primary and the secondary flows can be represented and quantified by distribution of Turbulent Kinetic Energy (TKE) in the flow domain of subject pump. The profile of non dimensional Tubulent Kinetic Energy along the centreline of the ejector pump, Figure 12, illustrates that immediately downstream of primary nozzle exit, their lies a low TKE region. This region corresponds to the unmixed high velocity potential core where the turbulence is minimal. Along the centreline as the strength of potential core diminishes, the growth rate of TKE increases and reaches its peak where the shear layers originated from the rim of primary nozzle exit, interact with each other. This is the region where the potential core vanishes. The TKE of the flow is then dissipated in the mixing / entrainment process of secondary fluid into the primary fluid as the large scale energy carrying eddies breaks down into smaller ones and hence the magnitude of TKE begins to decrease at further downstream and eventually it vanishes out. This captured phenomena is very much in agreement with the literature.

CONCLUSION

The application and efficacy of Embedded Large Eddy Simulation, a hybrid RANS-LES scheme, for flow field visualization of ejector pump used in aircraft fuel system is studied in this research using ANSYS Fluent® software. The numerically simulated patterns of flow field characteristics and the physics of flow using Embedded LES technique are in good agreement with the literature. As this scheme incorporates the advantages of both the RANS and LES turbulence models, the cautious application provides the information regarding flow instabilities which is not possible by using the RANS turbulence model approach alone. The primacy of LES approach in visualization and analysis of complex flow field of ejector pump is fully explored while at the same time, the associated computational cost is also reduced as compared to pure LES.

NOMENCLATURE

μ	Laminar viscosity
μ_{sgs}	LES Subgrid Scale viscosity
S	Strain rate
Ω	Vorticity rate
ε	Turbulent energy dissipation rate

k	Turbulent kinetic energy
L_t	Turbulent length scale
l_o	Integral length scale
ρ	Density
Un	Mean flow velocity at primary nozzle exit
Ps	Secondary flow inlet pressure

REFERENCES

[1] B. Adrienne, Y. Bartosiewicz and G. Sirinivas, 2015,"Visualization and Validation of Ejector Flow Field with Computaional and First-Principles Analysis," *ASME Journal of Fluid Engineering,* vol. 137(5) 051107.

[2] J. H. Im and S. J. Song,2015, "Mixing and Entrainment Characteristics in Circular Short Ejectors," *ASME Journal of Fluids Engineering,* vol. 137(5) 051103.

[3] P. Desevaux, A. Mellal and Y. A. d. Sousa,2004,"Visualization of Secondary Flow Choking Phenomena in a Supersonic Air Ejector," *Journal of Visualization,* pp. 249-256.

[4] J. Hoke, R. Bradley, J. Stutrud and F. Schauer,2002,"Integration Of A Pulsed Detonation Engine With An Ejector Pump And With A Turbocharger As Methods To Self-Aspirate," in *40th AIAA Aerospace Sciences Meeting & Exhibit*, Reno, NV.

[5] W. H. Kim and T. S. Park,2013, "Effects of Noncircular Inlet on the flow structures in Turbulent Jets," *Journal of Applied Mathematics and Physics*, vol. 1, pp. 37-42.

[6] Dennis A. Yoder et al,2013, "Modeling of Turbulent Free Shear Flows," in *43rd Fluid Dynamics Conference and Exhibit AIAA*, San Diego, California.

[7] J. Masud and M. Imran,2015, "Turbulence Modeling for Realistic Computation of Internal Flow in Liquid Ejector Pumps," in *53rd AIAA Aerospace Sciences Meeting.*

[8] J. Masud and A. Javed,2007, "Modification of Standard K-ε Turbulence Model for Analysis and Design Refinement of Fuel Ejector Pump," in *45th AIAA Aerospace Sciences Meeting* , Nevada, USA.

[9] A. Bouhanguel, P. Desevaux and E. Gavignet,2015, "Visualization of flow instabilities in supersonic ejectors using Large Eddy Simulation," *Journal of Visualization*.

[10] F. Menter,2011, "Turbulence Modeling for Engineering Flows," ANSYS, INC.

[11] D. Cokljat, D. Caradi, G. Link, R. Lechner and F. Menter, 2009,"Embedded LES Methodology for General Purpose CFD Solvers," in *6th Int. Symp. Turbulence and Shear Flow Phenomena,,* Seoul,

Korea.

[12] F. Mathey, D. Cokljat, J. Bertoglio and E. Sergent,2003,"Specification of LES Inlet Boundary Condition Using Vortex Method," in *4th International Symposium on Turbulence, Heat and Mass Transfer*, Antalya.

[13] I. Celik, Z. Cehreli and I. Yavuz,2005, "Index of resolution quality for large eddy simulations," *Journal of Fluids Engineering*, vol. 5, no. 127, p. 949–958.

[14] F. M. White,1999, *Fluid Mechanics*, McGraw-Hill.

[15] G. Aleksey, 206, "Quick guide to setting up LES-type Simulations," ANSYS Inc, Sweden.

[16] K. Aldas and R. Yapici, 2014,"Investigation of effects of scale and surface roughness on efficiency of water jet pumps using CFD," *Engineering Applications of Computational Fluid Mechanics*, vol. 8(1), pp. 14-25.

[17] T. Karimipanah, 1996, "Turbulent jets in confined spaces," Centre for Built Environment, Royal Institute of Technology, Sweden.

LIST OF FIGURES

LIST OF TABLES

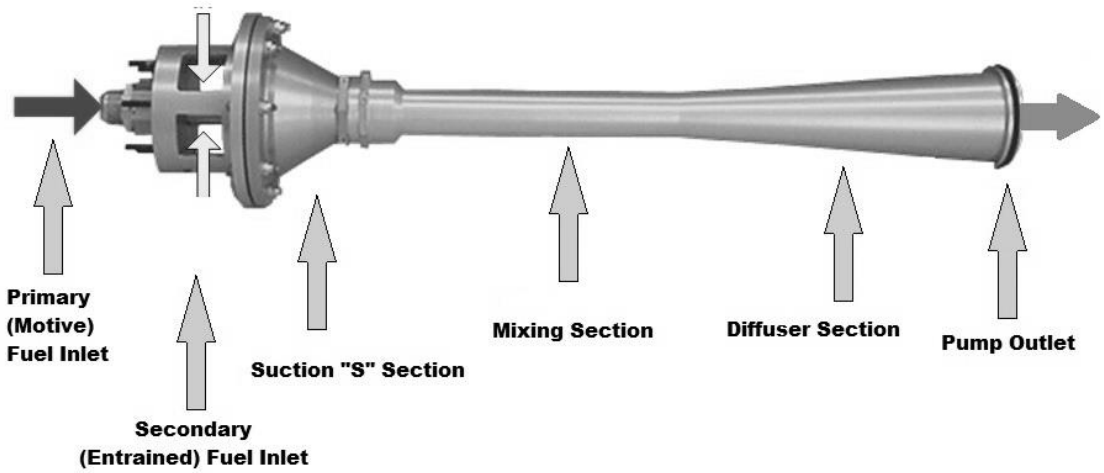


Figure 1 : Different sections of Ejector Pump

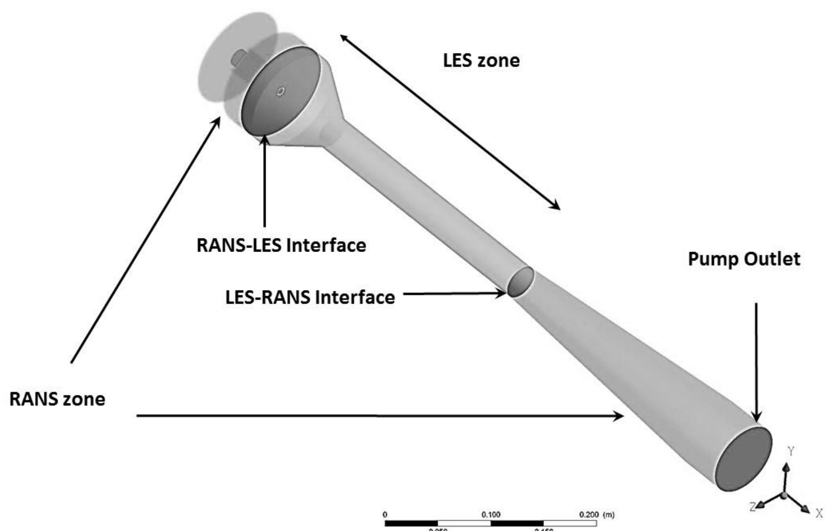


Figure 2 : 3D CAD model of ejector pump showing various sections and interfaces

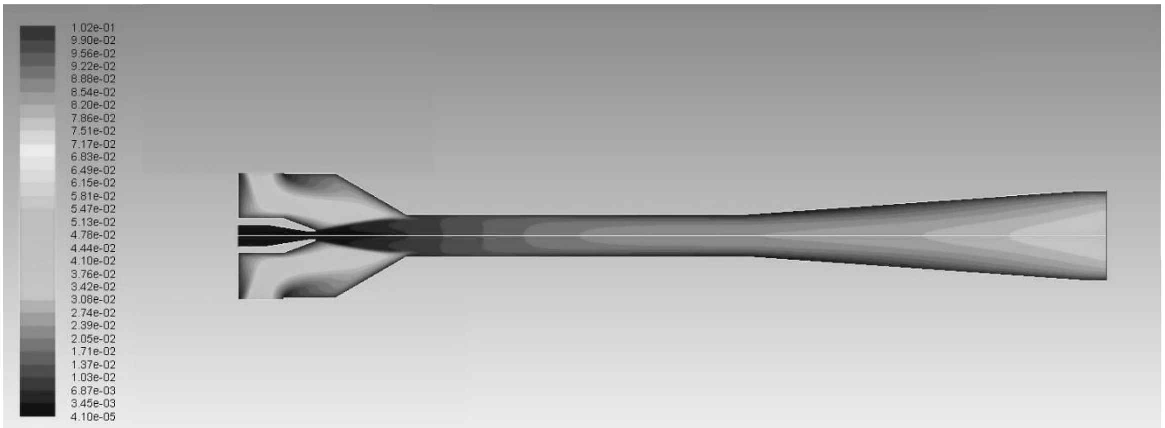
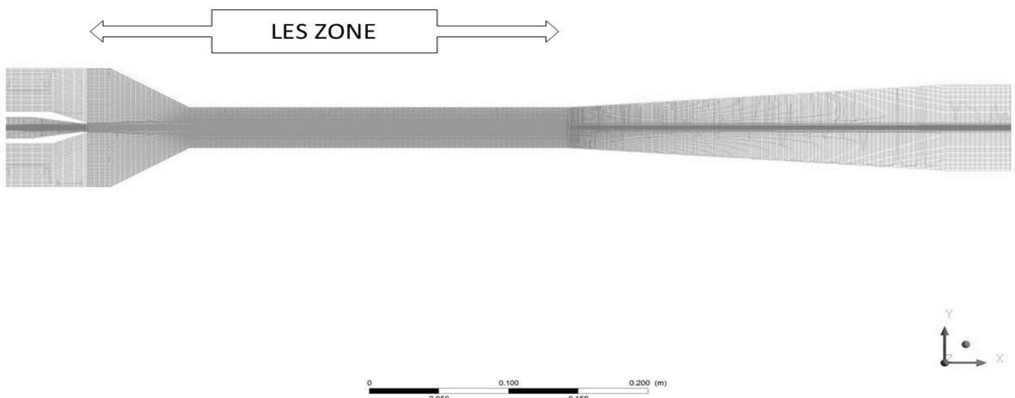
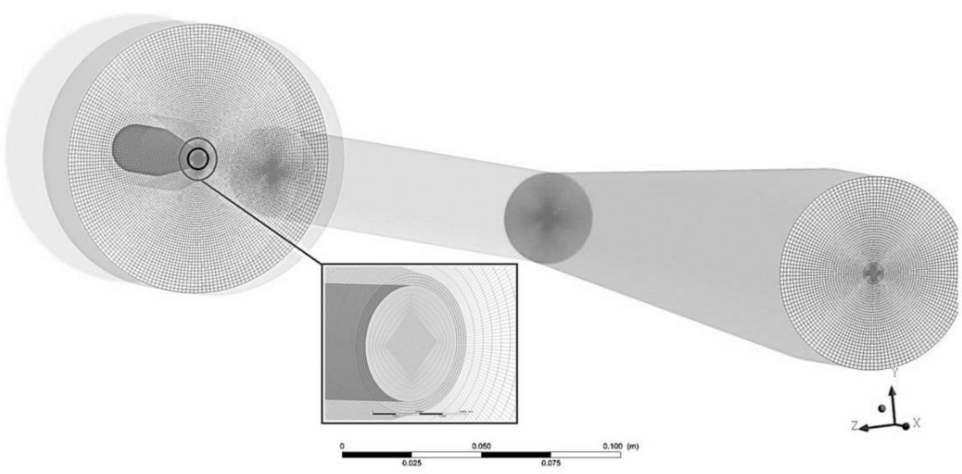


Figure 3 : Contours of Integral Length Scale in LES zone



(a)



(b)

Figure 4 : Mesh distribution inside ejector pump (a) cross sectional view (b) meshing at different cross sections i.e. RANS-LES interface, Mixing section inlet & exit and Pump outlet

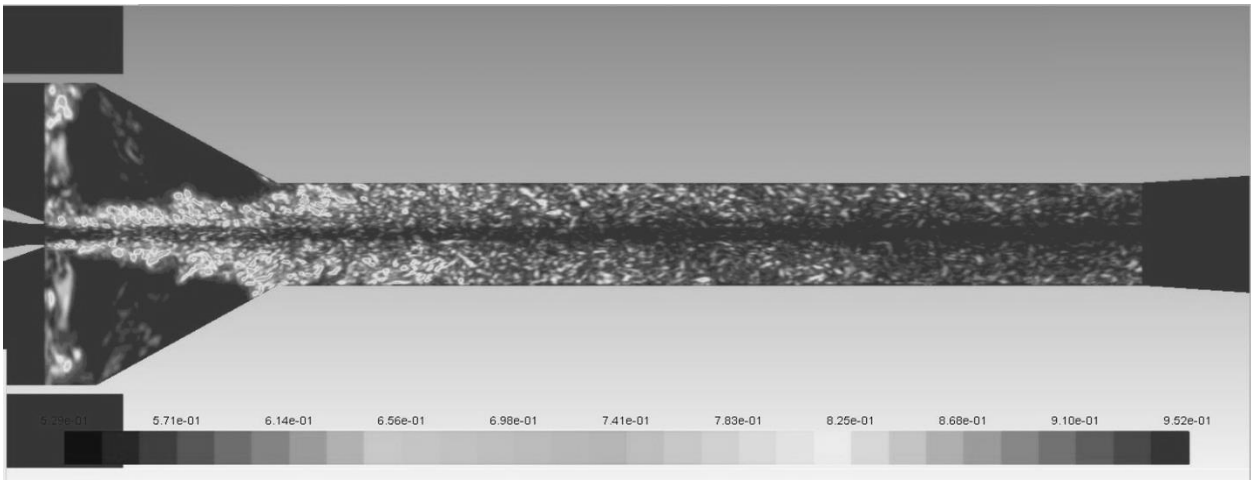


Figure 5 : Contour plot of LES quality index

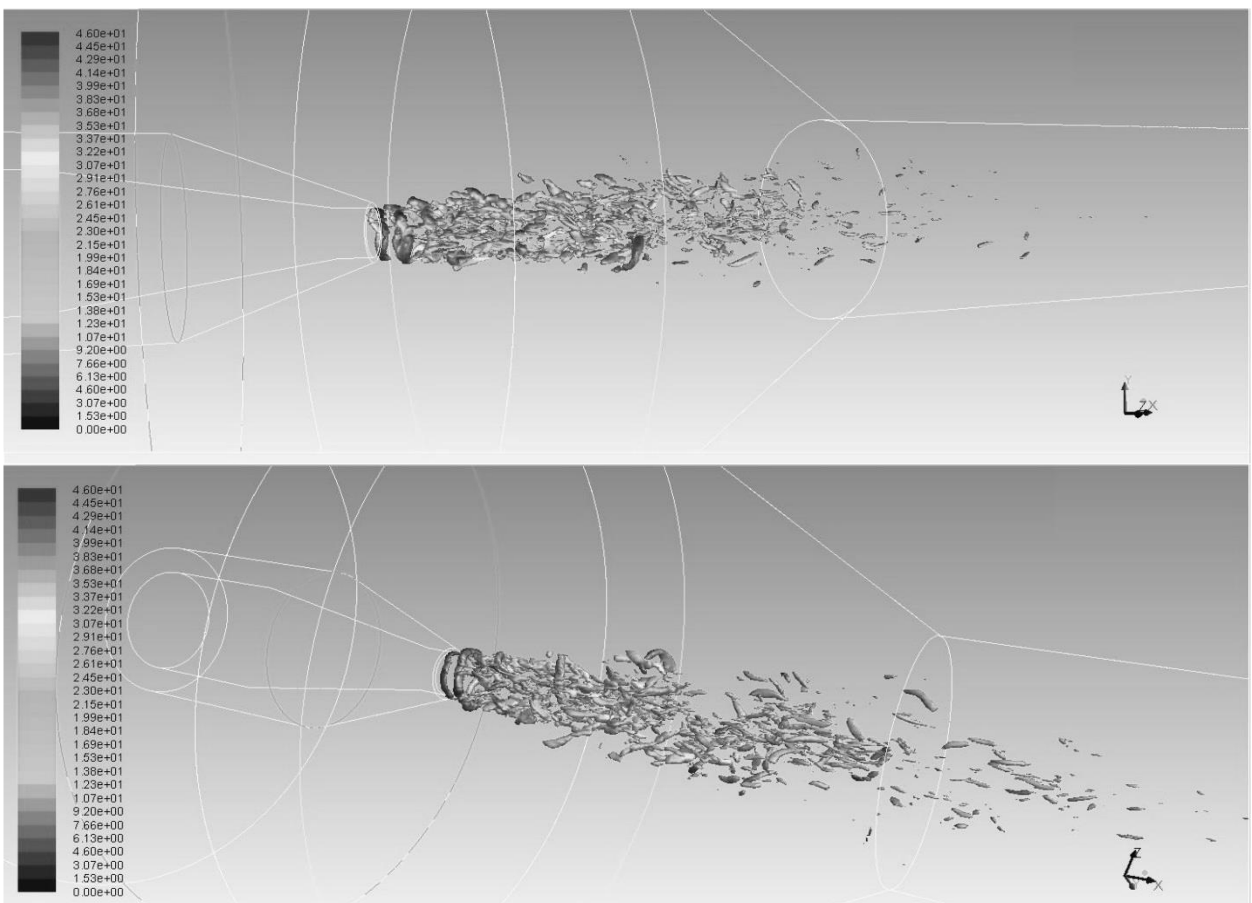


Figure 6 : Iso contour of Q-criterion colored with velocity magnitude

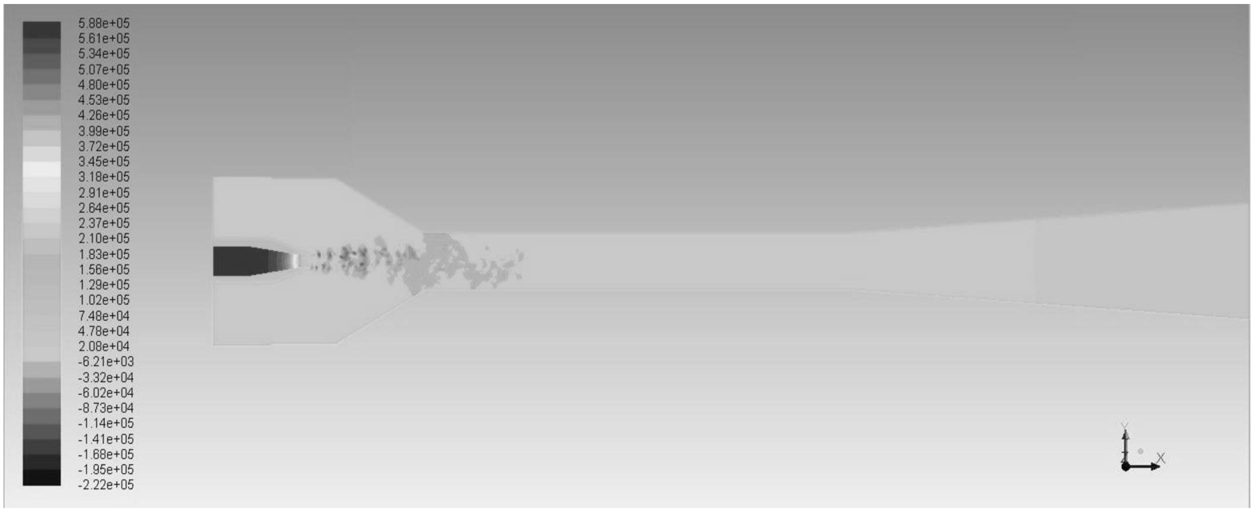


Figure 7 : Instantaneous Pressure Field of ejector pump

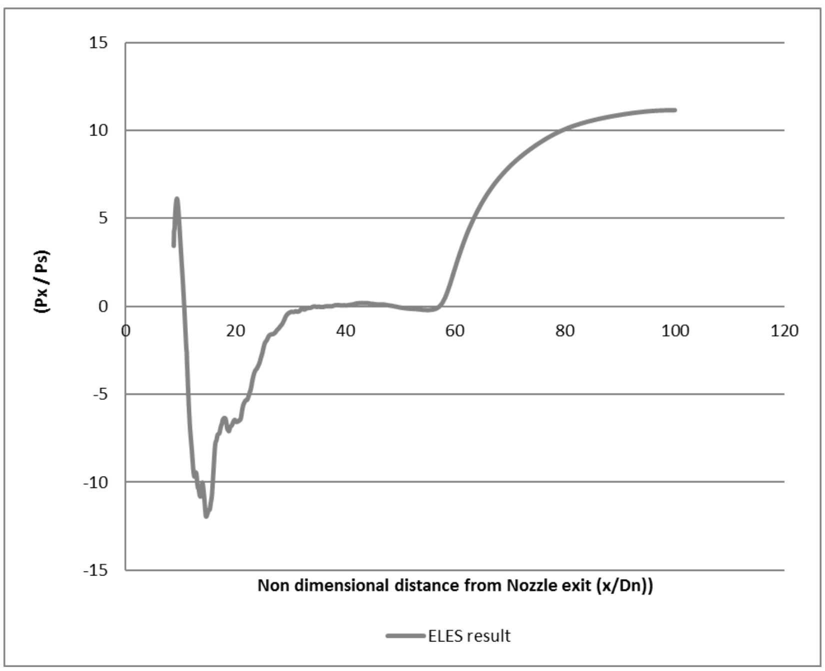


Figure 8 : Variation of mean static pressure along the centerline of ejector pump

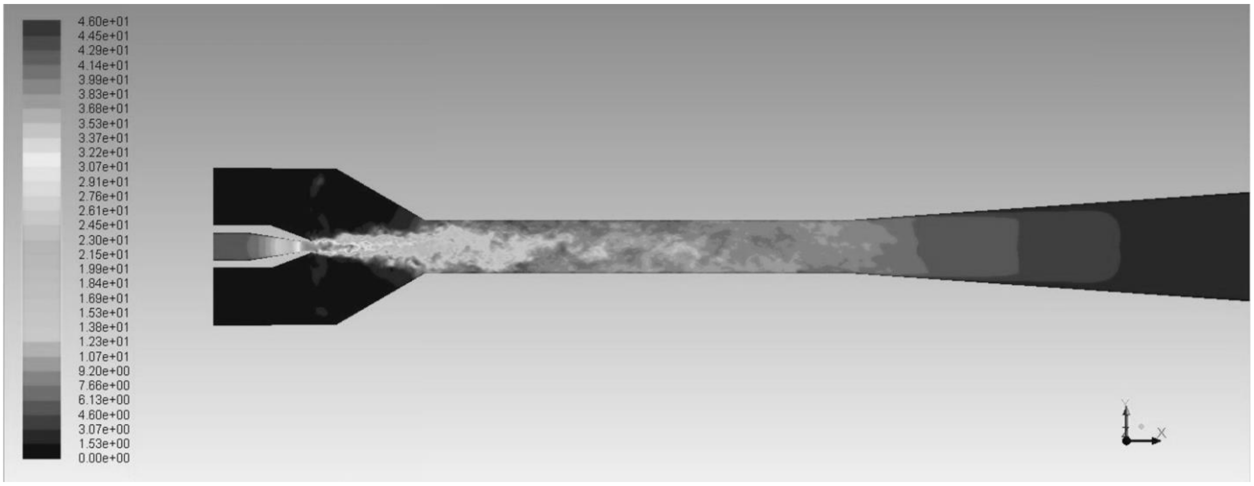


Figure 9 : Instantaneous velocity field of ejector pump

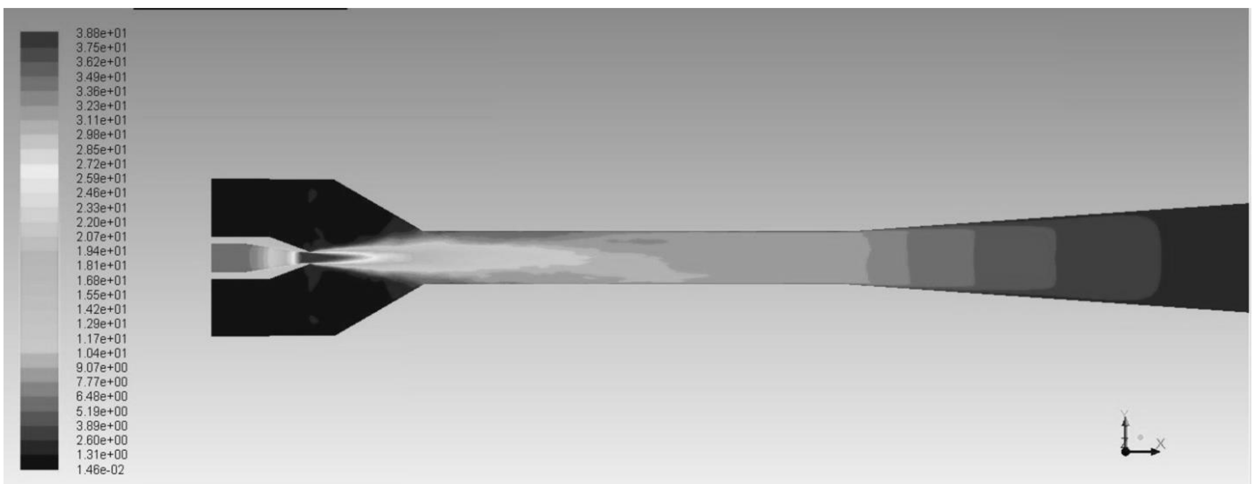


Figure 10 : Mean Velocity Magnitude contours at the cross section of ejector pump

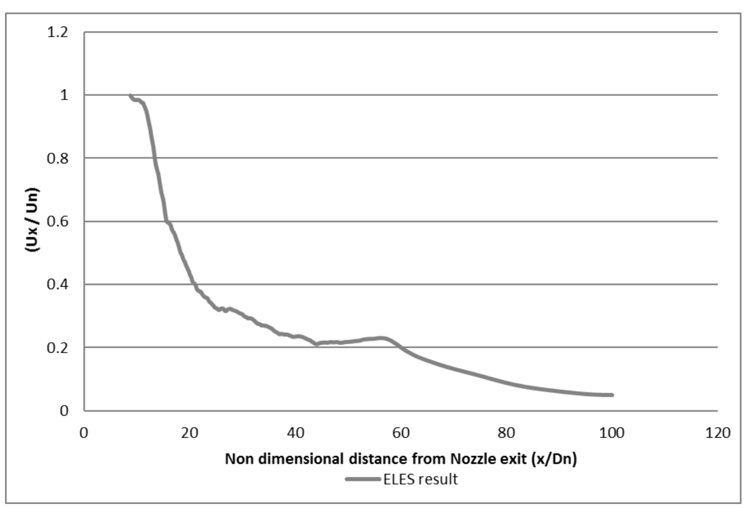


Figure 11 : Mean Velocity Profile along the centerline of ejector pump

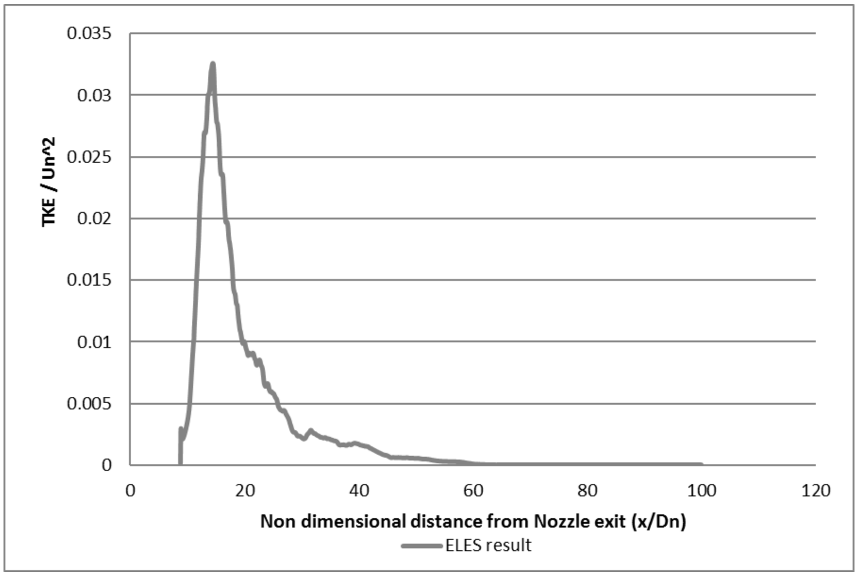


Figure 12 : Variation of Turbulent Kinetic Energy (k) along the centerline of ejector pump

Table 1 : Comparison of experimental and numerical results

Inlet press (MPa)		**Pump Outlet press (MPa)**	**Pump outlet mass flow rate (kg/s)**		
Pri Nozz	**Sec Nozz**		**Exp**	**ELES**	**% diff**
0.60000	0.00224	0.02500	5.933	5.944	**+0.011**
1.20000	0.00224	0.05100	8.372	8.4676	**+0.095**
1.80000	0.00224	0.07800	10.158	10.358	**+0.200**