INTERNAL FLOW IN A Y-JET ATOMISER ---NUMERICAL MODELLING---

Z. Tapia, A. Chávez
e-mail: **ztapia@imp.mx**
Instituto Mexicano del Petróleo
Blvd. Adolfo Ruiz Cortines 1517-12, Fraccionamiento Costa de Oro,
94299, Boca del Río, Veracruz, México.
Phone: 52-229-9222851, Fax: 52-229-9222858

Abstract:
A numerical modelling of the internal flow through a single two-dimensional port of a Y-jet atomizer was done. The gas-liquid flow interaction inside the mixing chamber of the atomizer was calculated by use of the computational commercial code FLUENT. Computational fluid dynamics simulations were performed in a three-dimensional grid representing the internal geometry of the atomizer. Four Y-jet atomiser configurations were modelled at one operating condition (40 kg/h oil and 1.71 kg/h air). For each atomiser the same geometry and dimensions were used except that the angle formed between the gas and liquid inlets was changed at 30, 50, 70 and 90 degrees. The inlets and outlet have a rectangular cross-section. The cross sectional area of the inlet and outlet channels was kept constant at 4.5 mm^2 for air, 6 mm^2 for oil and 7.5 mm^2 for the outlet port. This particular geometry was selected in order to make a qualitative comparison of the computational results with those of a former experimental study on an atomiser with the same shape and dimensions. The modelling of all the cases revealed that the available energy of the atomising fluid is not fully used in the acceleration and break-up of the oil stream. From the entry angles modelled, it is apparent that the entry angles above 50 degrees produce the best mixing of the oil and air. From this knowledge, it can be concluded that finer modifications of the internal geometry and wall texture are needed in Y-jet atomisers in order to improve the atomisation process.

Introduction

Y-jet atomisers (figure 1) are extensively used in oil fired utility boilers. Together with the burner aerodynamics, the atomisers represent the core of the combustion system and their good performance plays a major role to keep a low level of pollutant formation and to maintain the combustion efficiency as highest as possible. The present study has the general purpose of improving the basic understanding of the flow structure that is formed in a Y-jet atomiser when the atomised and atomising fluids come into contact. To accomplish this objective, a numerical modelling of the structure and dynamics of the internal mixing flow has been done. It is felt that an understanding of the basic jet atomisation phenomenon gained in this way would be very helpful for the understanding of the more complicated jet disintegration process of practical injection systems. Four Y-jet atomiser configurations were modelled at one operating condition (40 kg/h oil and 1.71 kg/h air). The atomisers have entry angles between the oil and air ports of 90, 70, 50, and 30 degrees. By varying the inlet angle it is possible to analyse the effects of the two interacting streams within the atomiser under different operating conditions. For the sake of brevity, only the geometry of the atomiser with an entry angle of 50 degrees is shown in figure 2. The air stream is introduced in the inlet 1 and the oil stream is introduced in the inlet 2. The inlets and outlet have a rectangular cross-section. This particular geometry used in the atomiser modelling was the same geometry used on a former experimental study [1].

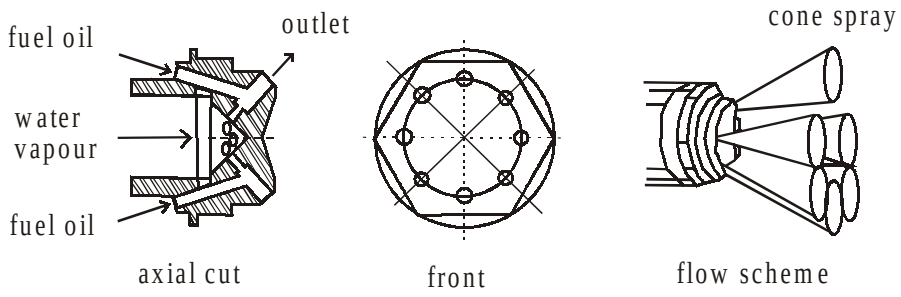


Figure 1. Prototype of a Y-jet atomiser.

The analysis of the internal flow in the mixing chamber of the atomiser is a highly complex problem due to the gas-liquid two-phase flow interactions, which cause violent mixing of the liquid and gas. Because of this difficulty, in the past the majority of investigators [2, 3] have focused their work on simulating the spray. The modelling of the internal flow in twin-fluid atomisers is practically non-existent. The characteristics of the two-phase flow involved in the physical atomisation process make the mathematical simulation very complex. In this work, mathematical modelling was carried out using a computational fluid dynamics (CFD) code called FLUENT [4]. In FLUENT, different models are available for the simulation of the overall physics of the problem. Essentially, the models for the flow interactions involve the solutions of mass and momentum conservation equations coupled to a Volume of Fluid (VOF) model, which is designed for two or more inmiscible fluids, where the position of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids is tracked throughout the domain. In the present work, the standard k-ε model of turbulence was used to take into account the interactions of the fluids with the walls.

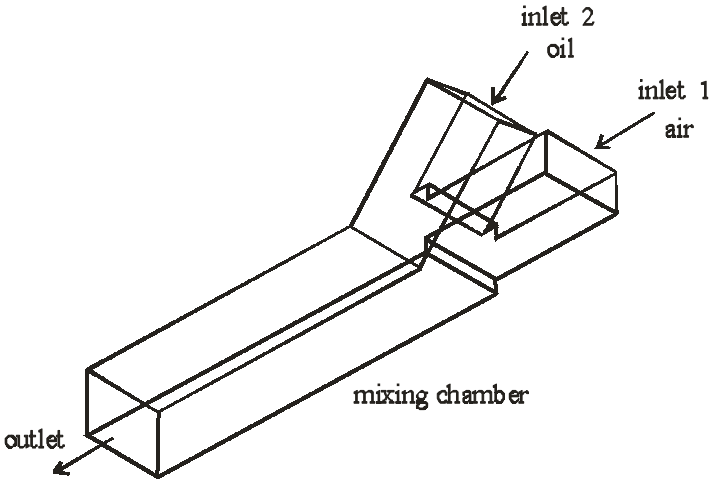


Figure 2. Atomiser geometry.

Computational Modelling of a Y-Jet Atomiser

Computational fluid dynamics essentially comprises the physical laws, which govern the interactions between the fluid elements in a flow. The physical laws determine the conditions at each position in the flow as a function of time. The governing equations representing the conservation of mass and momentum in Cartesian coordinates assume the following form when written in tensor notation:
Equation of continuity (mass conservation):

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \tag{1}$$

Momentum conservation equation (Navier-Stokes equation):

$$\frac{\partial}{\partial t}(\rho u_j)+\frac{\partial}{\partial x_i}(\rho u_i u_j)=\frac{\partial}{\partial x_i}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]-\frac{\partial p}{\partial x_j}+\rho g_j \tag{2}$$

Here the left hand side represents convection, and the first term on the right hand side represents diffusion. The remaining terms represent pressure, and the body force of gravity.

The above equations are transformed into a set of algebraic equations that can be solved iteratively. In this transformation, also called discretisation, the domain is assumed to be divided into cells or control volumes, and the governing equations are integrated over each cell. To carry out the discretisation of the governing equations, the physical space is subdivided into a number of cells by means of a structured grid. The cells generated are defined by the points where the grid lines intersect (node points). The discretisation procedure consists of the integration of the conservation equations over each control volume assuming uniform values of the variables over the cell. From this procedure, one obtains a set of algebraic equations, which relate the values of the variables at

the centres of the cells and at the faces of the cell, the fluxes across the faces of the cell and the dimensions of the cell. Then this set of algebraic equations is solved by the code.

The VOF formulation relies on the fact that the fluid phases are not interpenetrating. In each control volume, the volume fractions of each phase add up to unity. The fields for all variables and properties are shared by the phases, as long as the volume fraction of each of the phases is known at each location. Thus, the variables and properties in any given cell are either purely representative of one of the phases, or are representative of a mixture of the phases, depending upon the volume fraction values. For example, in the particular case of this study, liquid and air are involved in the process. If the volume fraction of the oil is denoted by ε_l, then the following three conditions are possible:

$\varepsilon_l = 0$	the cell is empty of liquid
$\varepsilon_l = 1$	the cell is full of liquid
$0 < \varepsilon_l < 1$	the cell contains the interface between the air and liquid

Based on the local value of the volume fraction of one of the fluids in a multi-fluid system, the appropriate properties and variables will be assigned to each control volume within the domain.

The tracking of the interface between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. For the k^{th} phase, this equation has the form:

$$\frac{\partial \varepsilon_k}{\partial t} + u_j \frac{\partial \varepsilon_k}{\partial x_i} = S_{\varepsilon k} \tag{3}$$

The source term on the right hand side of the equation (3) is normally zero. ε_k is the volume fraction of the k^{th} fluid in a multi-fluid system.

The properties appearing in the transport equations are determined by the presence of the component phases in each control volume. In the particular case of this work, a two-phase system is being studied. If the volume fraction of the liquid phase is being tracked, the density in each cell is given by:

$$\rho = \varepsilon_l \rho_l + (1 - \varepsilon_l)\rho_a \tag{4}$$

The viscosity is computed in the same manner.

The momentum equation (2) is solved throughout the domain, and the resulting velocity is shared among the phases. The momentum equation is dependent on the volume fraction of any phase through the properties ρ and μ. The VOF formulation in FLUENT is time-dependent, so the continuity and momentum equations are solved using an explicit time-marching scheme.

Before beginning the full three-dimensional computations, a simulation was first made in two dimensions using 145x57 cells. The grid used in two dimensions was selected as finer as possible in order to evaluate the effect of the spatial resolution. The three dimensional grid had 73x29x8 cells. The computational grid of the atomiser with an entry angle of 50 degrees is shown in figure 3. The grids used for the other entry angles have a similar distribution and number of cells and are, therefore not shown.

The inlet boundary conditions (such as the turbulence intensity and dissipation length scale) used in the atomiser modelling are presented in Table 1. The velocity boundary conditions were uniform at each inlet; in the direction of the inlet channel. The physical properties of the liquid and the air used for the modelling are shown in Table 2.

Inlet	Fluid	Volume fraction	Turbulence intensity (%)	Characteristic length (m)
1	Air	0	5	1.40×10^{-4}
2	Oil	1	0.1	1.68×10^{-4}

Table 1. Boundary conditions.

	Density (kg/m^3)	Dynamic viscosity (kg/m s)
Air	1.2	1.72×10^{-5}
Oil	909.6	2.72×10^{-2}

Table 2. Physical properties.

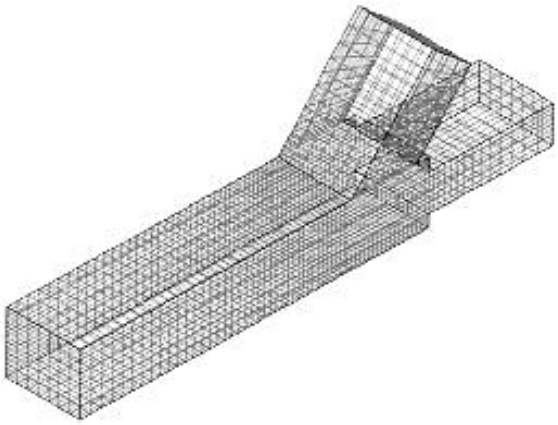
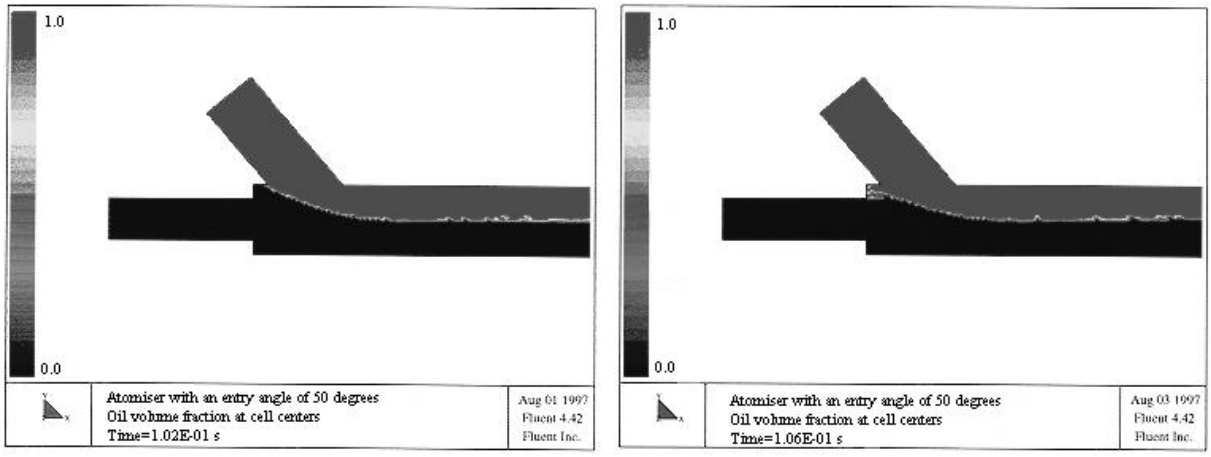


Figure 3. The grid used in the computation.

The simulation in the atomiser with an entry angle of 50 degrees for the case of two dimensions is presented in figure 4. This figure shows the variation of the air-liquid interface at two time intervals. The dark zone corresponds to the air stream and the dim zone that is lighter than the dark zone corresponds to the oil stream. The oil and air flow rates were 40 and 1.71 kg/h, respectively. These flow rates correspond to bulk velocities at the inlets of 2.04 m/s for the oil and 87 m/s for the air. In Figure 4 (b), it is observed that a small disturbance starts to develop on the liquid surface where the two streams start to mixing. Also it is observed that the small disturbance grows in size and changes its position with time, but no liquid is found to detach from the main body of liquid in the simulation.



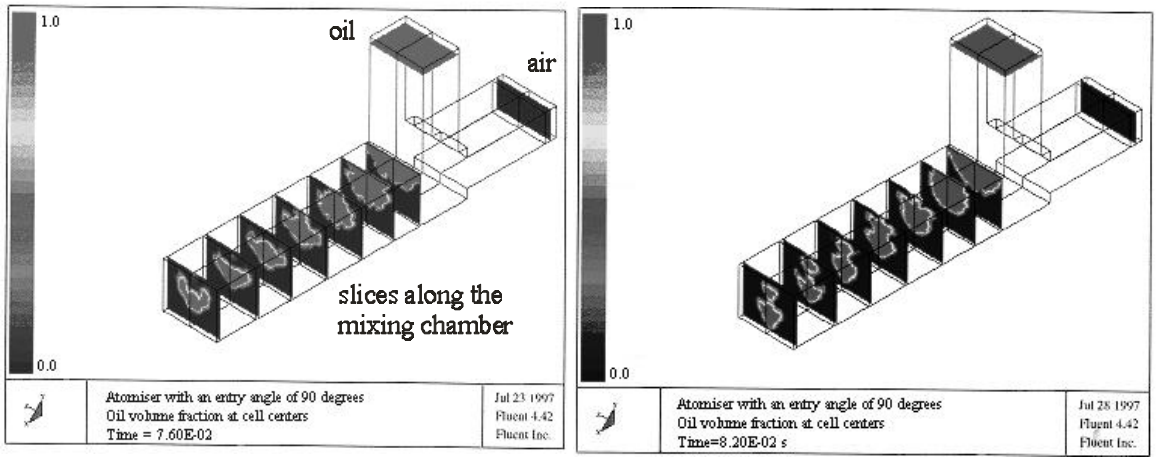
(a) (b)

Figure 4. Computational results of the 2-D case at different time (a, b), 40 kg/h oil, 1.71 kg/h air.

Figures 5, 6, 7 and 8 show the oil volume fraction for the four entry angle modelled for the three dimensional case. The results presented are for the same set of operating conditions as the two-dimensional computations used. Two consecutive times at intervals of 6 ms are shown for each inlet angle. The development of the internal flow can be clearly observed from the figures. Seven slices along the mixing chamber are presented for each simulation. It is observed a complete detachment of the liquid stream beginning near the oil entry position. Another characteristic that is observed in the figures is that the air stream forms an approximately annular flow around the oil stream. Large differences in the shape of the oil core are found. Indeed, in some of the figures (at some times) part of the oil remains on the top wall even at the positions furthest downstream. Also, it was observed that the shape of the interface changed both along the mixing chamber and with time. One characteristic that also was noted is that the liquid stream tends to detach from the wall that corresponds to the oil inlet port. However, this detachment only happens for angles bigger than 30 degrees. For the entry angle of 30 degrees, the oil stream entering the mixing chamber cannot penetrate fully the air stream and is forced toward one side of the mixing chamber.

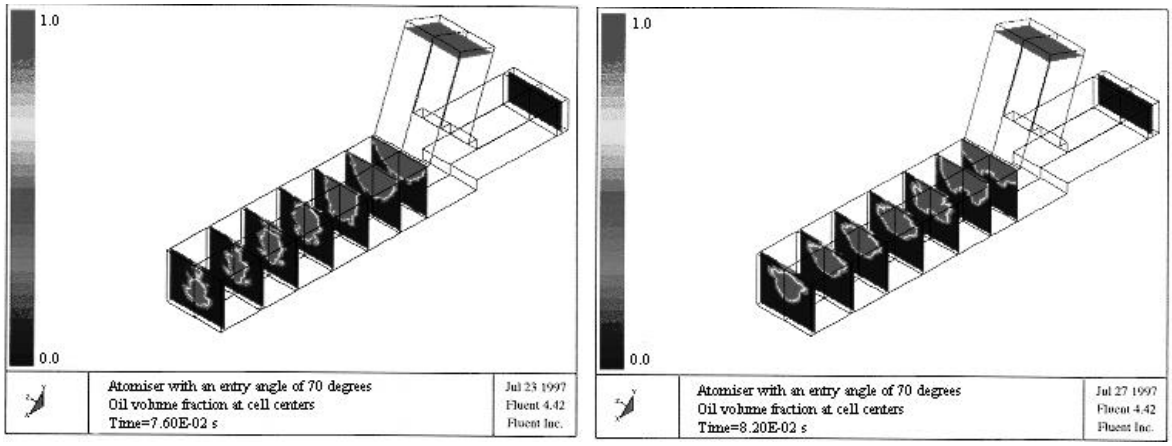
From the images presented in a former work [1] revealed visual evidence of the variation of the air/liquid interface inside the mixing chamber of the Y-jet atomiser tested. The visualisation and computational results illustrate that there is a tendency for the liquid to attach to one side of the mixing chamber, with the consequence that

the sprays produced for this type of atomiser are characterised by poor symmetry. This asymmetry distribution of the liquid is the cause that coarse droplets are formed in the spray edge of Y-jet atomisers.



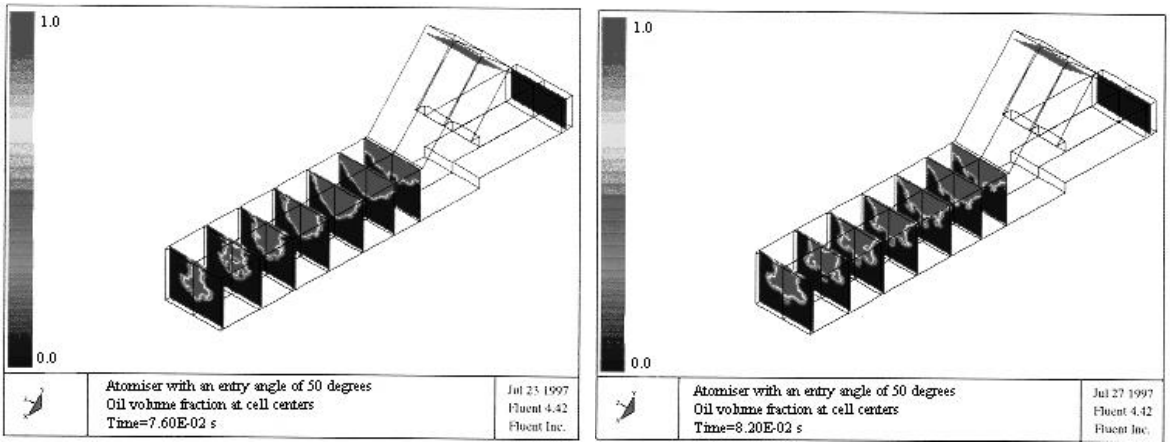
(a) (b)

Figure 5. Computational results of the 3-D case for the 90 degrees entry angle at different time.



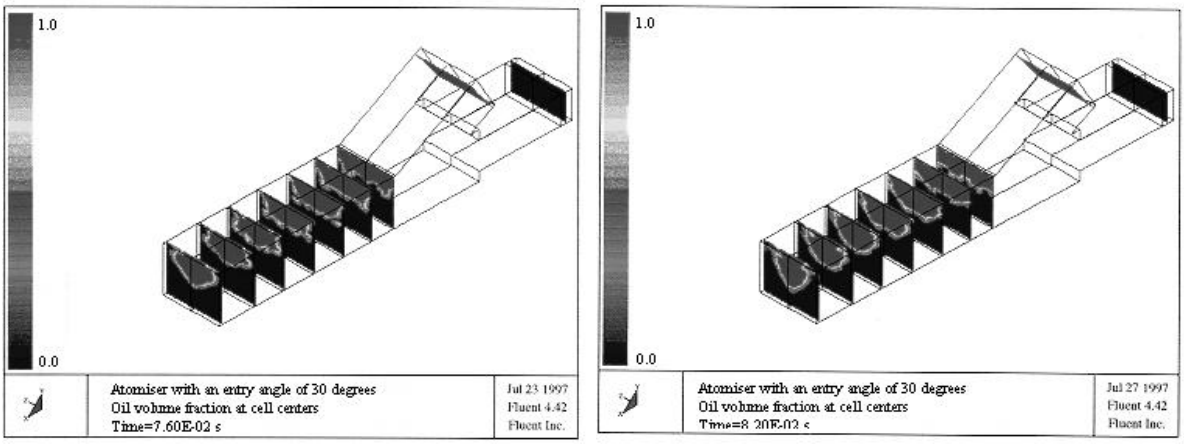
(a) (b)

Figure 6. Computational results of the 3-D case for the 70 degrees entry angle at different time.



(a) (b)

Figure 7. Computational results of the 3-D case for the 50 degrees entry angle at different time.



(a) (b)

Figure 8. Computational results of the 3-D case for the 30 degrees entry angle at different time.

Conclusions

A numerical modelling of the structure and dynamics of the internal mixing flow in a Y-jet atomiser has been done. It is felt that an understanding of the basic jet atomisation phenomenon gained in this way would be very helpful for the understanding of the more complicated jet disintegration process of practical injection systems. It was noted that the oil stream entering the mixing chamber cannot penetrate fully the air stream and is forced toward one side of the mixing chamber. Because of the mal-distribution, the energy of the atomising fluid is not fully used in the acceleration and break-up of the oil stream. From the computational results, it was noted that the air stream forms an annulus around the liquid stream. Also, it was noted that the shape of the interface changed both along the mixing chamber and with time. One characteristic that also was noted is that the liquid stream tends to detach from the wall that corresponds to the oil inlet port. When the inlet angle decreases, the tendency of the oil stream to remain attached to the oil inlet wall increases. Further work it is necessary in order to use the energy of the atomising fluid in a more efficient way. To improve the mixing between both fluids involved in the atomisation, some modifications on the internal dimensions of the atomiser are necessary to made as a future work.

Nomenclature

p	pressure [Pa]
x_i	position [m]
t	time [s]
u_i	velocity [m/s]
ε_l	volume fraction of the liquid [dimensionless]
ε_a	volume fraction of the air [dimensionless]
μ	dynamic viscosity [kg/m s]
ρ_l	density of the liquid [kg/m^3]
ρ_a	density of the air [kg/m^3]

References

[4] Z. Tapia, A. Chávez, J. Swithenbank and J.M. MacInnes, "Internal Flow in a Y-Jet Atomiser --Empirical Modelling –" *Procc. Of the 2nd International Symposium on Two-Phase Flow Modelling and Experimentation*, Vol. 3, Eds: G.P. Celata, P. Di Marco and R.K. Shah, Edizioni ETS, Pisa, Italy, 1999, pp. 1909-1914.

[2] J.M. MacInnes and F.V. Bracco, "Computational of the spray from an air-assisted fuel injector", *International Fuels and Lubricants Meeting and Exposition*, Tulsa, Oklahoma, USA, 1990.

[3] R.D. Reitz, *Atomisation and other breakup regimes of a liquid jet*, Ph.D. thesis, University of Princeton, USA, 1978.

[4] FLUENT EUROPE, Computational Fluid Dynamics software, user's guide, Creare Inc. Release 4.4, Lebanon, NH, USA, 1996.