Numerical Simulation of a Pseudo Plastic Fluid Through Sudden Enlargement

Djamel BELATRACHE[1], Abdelkader HARROUZ [1, *], A. ABDERRAHMANE[2], S. MANAA[1]

[1]*Laboratory of Sustainable Development and Computer Science (L.D.D.I), Department of Hydrocarbons and Renewable Energies, Ahmed Draia University - Adrar (Algeria).*

[2]*Department of Material Science, Adrar University, Adrar, Algeria.*

[*]*Corresponding author; Email: abd.harrouz@univ-adrar.dz*

Article Info

Article history:

Received 28 April, 2019
Revised 12 May, 2019
Accepted 28 May, 2019

Keywords:

Simulation
Modeling
Fluid
Orifice plate
Iso5167
CFX

ABSTRACT

This paper presents the flow of non-Newtonian fluids through sudden enlargements. The calculations are done by a code with the finished volumes. The stabilizing effect of the physical characteristic of the fluid is taken into consideration. In addition, we set as objective the influence of the main parameters like the index of structure of the fluid, the Reynolds number and the aspect ratio of the widening, on the evolution of the velocity profile, the length of establishment of the flow in front of the enlargement as well as on the recirculation zone. The results obtained were confronted whenever possible with results from other literature.

I. Introduction

Most fluids used in many industrial activities (Mechanical, Naval, Chemical) have a complex behavior their transport in the driving networks often include singularities (enlargements, contractions, valves, elbows). Most of the fluids encountered are non - Newtonian and the rheofluidifying character is found in many of these fluids and the consequences of this rheofluidification must be identified for a better prediction of the flow structure [1].

The objective of this research work is to carry out a purely digital investigation using high performance software (CFX) to correctly predict the velocity fields of a set of pseudo plastic fluids (rheofluidifier) through sudden enlargements [2]. Note that this type of software plays a very important role in the understanding of flows in singularities. It has become a very useful tool for very complex flows of fluids because the experimental study remains very expensive and sometimes impossible to perform full scale tests, therefore, a particular interest is given to the numerical prediction. Calculation codes and experimental validation improve the design of many systems designed for the transport of fluids.

II. Mathematical Modelling

An incompressible fluid is characterized by the fact that its density is constant in all the fields and does not vary with time and when its density does not depend on the pressure and depends only on the temperature. Generally, in fluid statics, liquids are considered incompressible fluid.

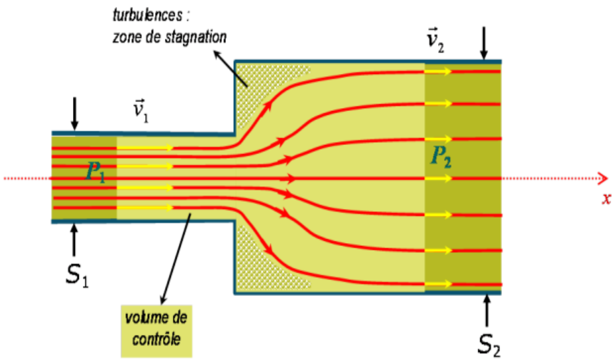

Figure 1. The sudden enlargement [3]

For a flowing fluid, this classification is reversed in a number of important situations. The sudden enlargement schematized in Figure 1, is the seat of pressure losses due to the presence of turbulence zones. They result from the abrupt change of direction imposed on the fluid, which under the effect of its own inertia, cannot follow closely the contours of the wall.

These zones of turbulence are therefore dissipative of energy but do not participate in the global flow since on average the speed is zero there: these zones are also called "zones of stagnation" [3-4]. The loss of charge generated by this singularity can then be evaluated analytically using the Euler theorem. To do this, consider as a control volume the space occupied by the fluid between the upstream section S_1 and the downstream section S_2. Let's also assume uniform velocities and pressures on these sections. So, we have:

$$\vec{R} + \vec{P} = q_m(\vec{v2} - \vec{v1}) \qquad (1)$$

That can be projected on a horizontal axis (that of the pipe) to obtain:

$$\vec{R}_x = q_m(\vec{v}_2 - \vec{v}_1) \qquad (2)$$

The resultant Rx of the surface forces acting along x is then constituted by the forces of pressure exerted on surfaces whose normal is oriented along x. We can therefore list:

- Upstream thrust + P1S1,
- Downstream thrust -P2S2,

The pressure force exerted on the annular surface S2 - S1 that, in fact, is the surface of the wall in contact with the stagnation zones. For this third force, we can therefore use the laws of hydrostatics and deduce that the pressure is exerted approximately the same as that which is exerted upstream, ie P1. Indeed, altitude differences being negligible, the pressure is almost uniform over the entire section S2 at the level of enlargement. Therefore, the third pressure force is expressed: + P1 (S2 - S1) [5].

We thus obtain:

$$\vec{R}_x = \vec{P}_1\vec{S}_1 - \vec{P}_2\vec{S}_2 + \vec{P}_1(\vec{S}_2 - \vec{S}_1) = q_m(\vec{v}_2 - \vec{v}_1) \qquad (3)$$

What is simplified in:

$$(\vec{P}_1 - \vec{P}_2)\vec{S}_2 = q_m(\vec{v}_2 - \vec{v}_1) \qquad (4)$$

By explaining the mass flow as a function of the flow velocity downstream, we have:

$$(\vec{P}_1 - \vec{P}_2)\vec{S}_2 = \rho\vec{v}_2\vec{S}_2(\vec{v}_2 - \vec{v}_1) \Rightarrow \vec{P}_1 - \vec{P}_2 = \rho\vec{v}_2(\vec{v}_2 - \vec{v}_1) \qquad (5)$$

Then, by a writing game, we can transform this equality by showing distinctly the static and kinematic pressures upstream and downstream:

$$\vec{P_1} + \frac{1}{2}\rho V_1^2 = P_2 + \frac{1}{2}\rho V_2^2 + \frac{1}{2}\rho(\vec{v_1} - \vec{v_2})^2 \qquad (6)$$

Where the last three terms of the right-hand side can be grouped together as a squared difference. This latter formulation shows, on the one hand, the total pressure upstream, on the other hand the total pressure downstream, and the difference between the two thus constituting the analytical expression of the pressure drop generated by the singularity [3 -5]. This pressure drop can also be formulated according to the kinetic pressure of the upstream flow: [6}

$$\Delta\vec{P_t} = \frac{1}{2}\rho(\vec{v_1} - \vec{v_2})^2 = \frac{1}{2}\rho(\vec{v_1} - \vec{v_1}\frac{\vec{S_1}}{\vec{S_2}})^2 = \frac{1}{2}\rho v_1^2(1 - \frac{\vec{S_1}}{\vec{S_2}})^2 \qquad (7)$$

It is thus possible to introduce a dimensionless coefficient, which depends solely on the geometry of the singularity, in this case upstream and downstream sections [7]:

$$\Delta\vec{P_t} = \frac{1}{2}\rho\vec{v_1}^2 K \text{ or } K = (1 - \frac{\vec{S_1}}{\vec{S_2}})^2 \qquad (8)$$

This result can be generalized to any type of singularity insofar as the singular pressure drop coefficient is known [8]. Under these conditions, for a complete hydraulic network comprising different sections of different lengths and sections and interconnected by singularities, it is possible to evaluate the total pressure losses between the input and the output of the circuit by formulating the generalized Bernoulli equation:

$$\vec{P_{tA}} - \vec{P_{tB}} = \Delta P_t = \sum_i \frac{1}{2}\rho\vec{v_i}^2\lambda_i\frac{L_i}{D_i} + \sum_i \frac{1}{2}\rho\vec{v_j}^2 K_j \qquad (9)$$

III. Simulation Results

The ICEM code gives two types of mesh: hexahedral and tetrahedral. For our study we used the mesh tetrahedral, the number of element changes according to the ratio of selected section: (for the case of the aspect ratio = 0.25 for example one generated a mesh of 296010 elements).

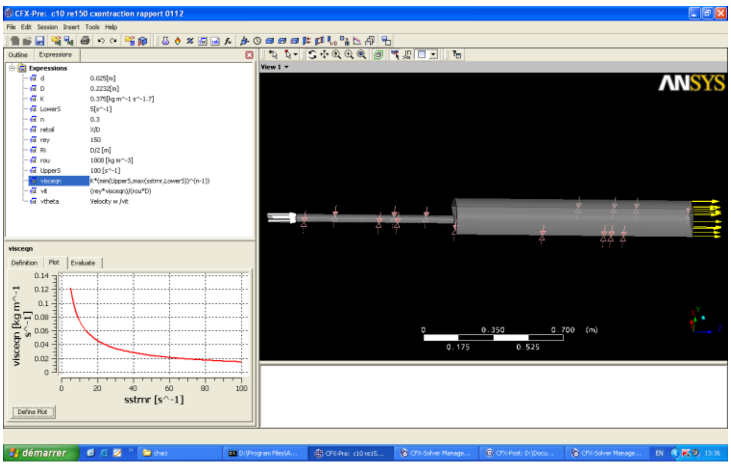

Figure 2. Screenshot of CFX-Pre

After the geometry drawing and the mesh generation, this geometry is exported to the CFX-Pre with the "Output" command.

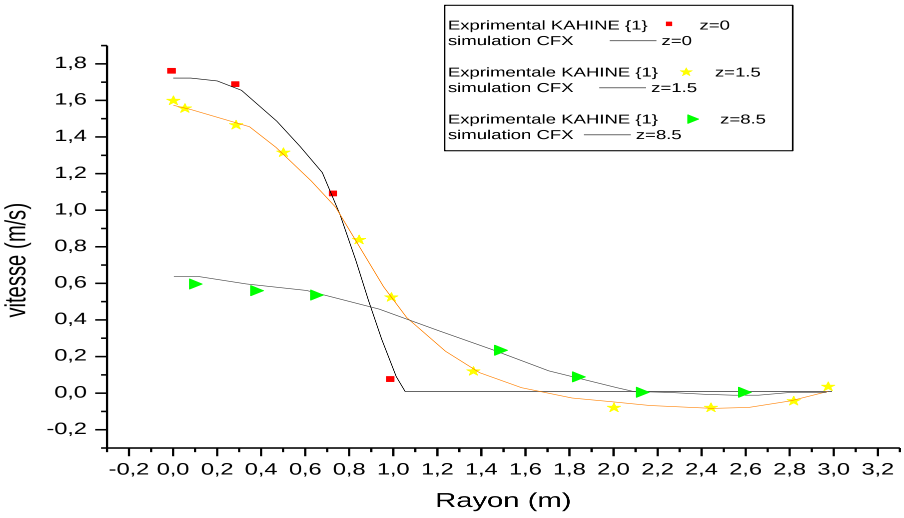

Figure 3. Validation of the axial speed profile

Figures 3 show experimental evaluations and simulation of the axial velocity profile downstream of the singularity, for different Reynolds number values. It is clear that as the Reynolds number increases, the vortex zone expands, the jet-to-wall bonding point moves downstream, and the flow-settling rate increases.

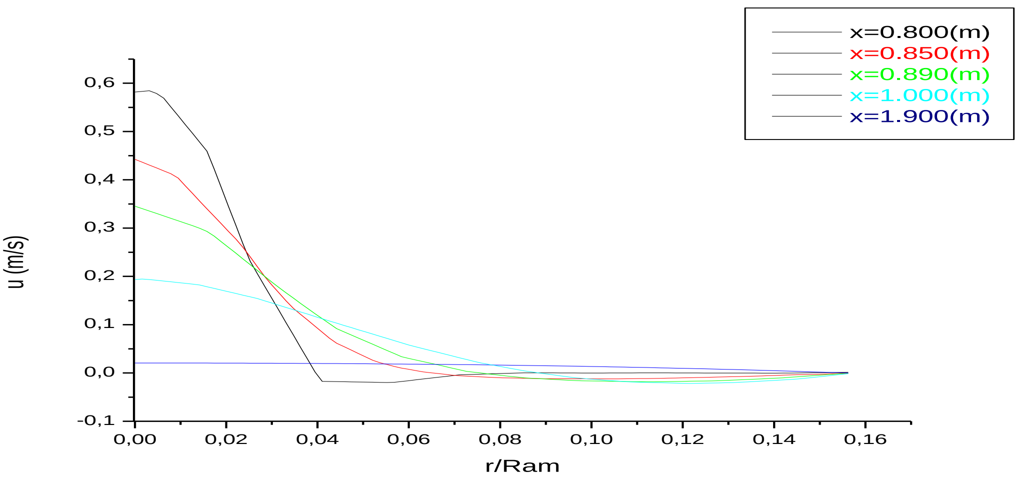

Figure 4. Validation experimental.

The results of figure 4 show a good agreement with experimental results of other author Dr. KAHINE of the University HENRI Poincare Nancy [1]. Nevertheless, with work considering the complexity of the flows as well as that of the code of computation; remains a small step in the field of flow simulation from where it will have to be exchanged by the study for example of a turbulent flow always thanks to the contribution of the CFX calculation code.

(a)

(b)

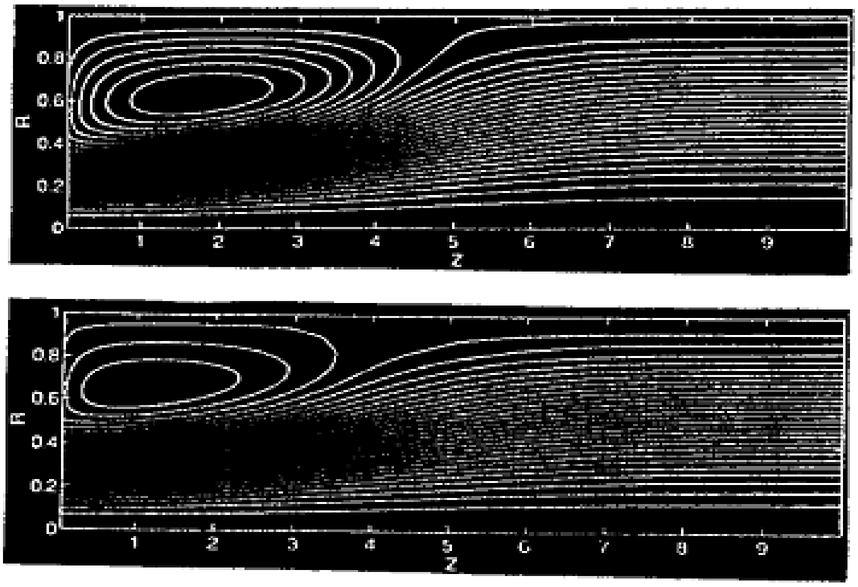

Figure 5. Influence of the aspect ratio on the evolution of the current lines:
(a) $S_{am} / S_{av} = 0.16$, (b) $S_{am} / S_{av} = 0.25$

In this part, the vortex zone is greatly reduced when the index of structure decreases. There is also a significant decrease in the establishment length when the rhea fluidizing character of the fluid increases.

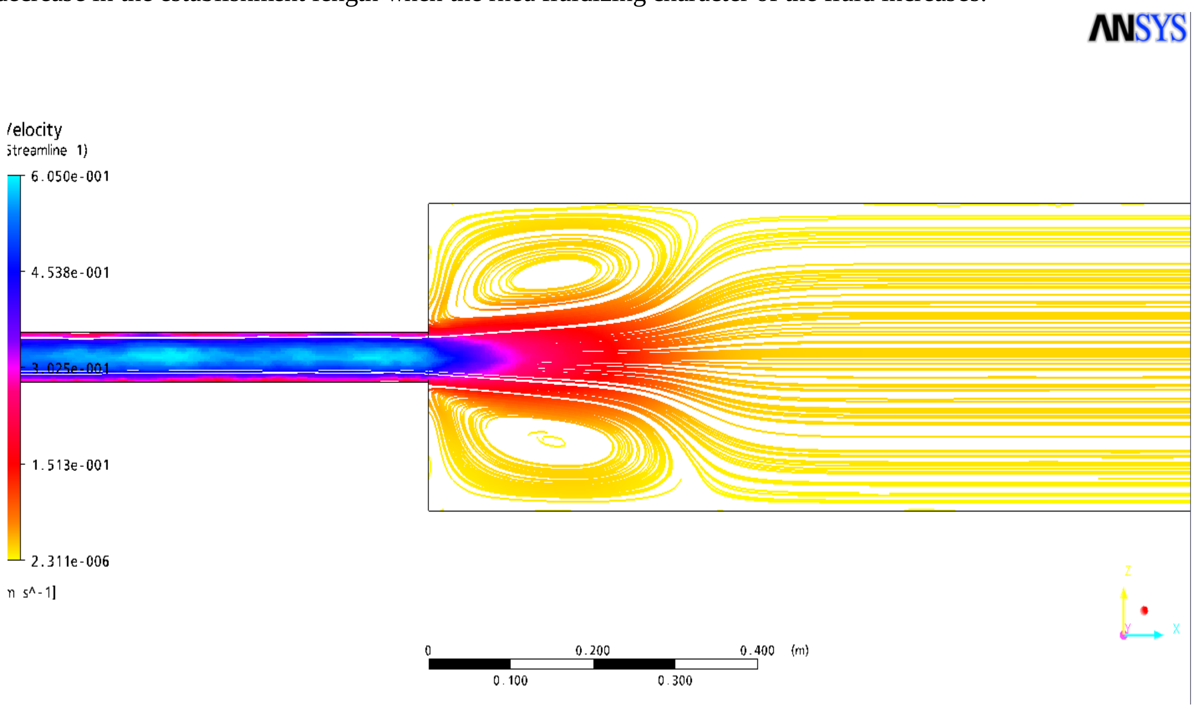

Figure 6. Influence of the aspect ratio on the evolution of the current lines:
$n = 0.6$, $S_{am} / S_{av} = 0.16$, $Re = 150$

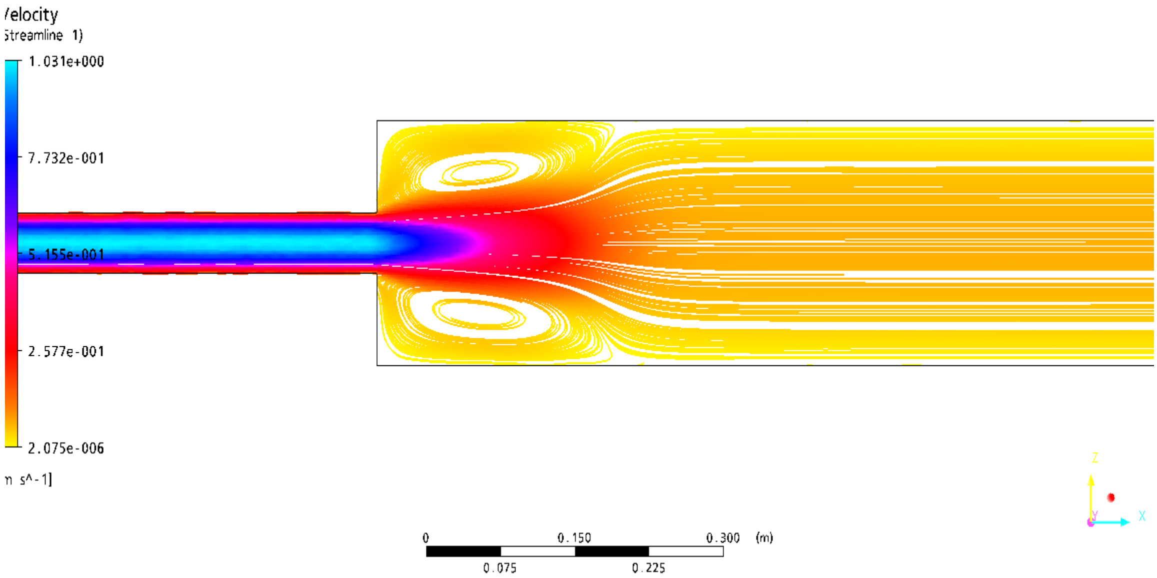

Figure 7. Influence of the aspect ratio on the evolution of the current lines:
$n = 0.6$, $S_{am} / S_{av} = 0.25$, $Re = 150$

Figures 6 and 7, showed the results of the data evolution of the current lines downstream of the enlargement, for different values of the aspect ratio. We notice that, when the aspect ratio decreases, the size of the swirling zone grows; the point of recollection point of the jet to the wall moves downstream and the length of establishment of the flow increases. In addition, we notice that there is a good agreement between our results and those found by Van Tuan Nguyen in Figure 7.

IV. Conclusion

This work presents the modeling of the flow of non-Newtonian pseudo plastic fluids through sudden expansions. The two-dimensional numerical simulation was realized thanks to a finite volume calculation code with which.
In this paper, we have set as objective the influence of the main parameters like the fluid structure index, the Reynolds number and the aspect ratio of the enlargement, on the evolution of the velocity profile, the length establishing the flow in front of the enlargement as well as on the recirculation zone. The results obtained confronted whenever possible with results from other literature.

Nomenclature

E	Energy
ρ	Density
g	Gravity Acceleration
H	Wave Height
T	Wave Period
J	Joule
N	Newton
m	Meter
s	Second
V	Volume
A	Area

References

[1] Kahine, « Ecoulement de Fluides non Newtoniens à Travers des Elargissements Brusques », Int. Comm. Heat. Mass. Transfert. Vol.24.N0 8, pp : 1103-1112 (1997).

[2] G. Ponthieu, “Ecoulement des Fluides Pseudoplastiques à travers des Elargissement Brusques », Thèse de doctorat Université Henri Poincaré Nancy 1 (1990).

[3] Van Tuan Nguyen et Michel Lebouche. Pertes de Charge des Fluides Non Newtoniens Thermo dépendants en Ecoulement Entre Deux Plaques Parallèles”, Comptes Rendus Mécanique 335, pp : 20 – 24 .(2007).

[4] R. Manica, “Simulation of Incompressible Non Newtonien Flows Throught Channel with Sudden Expansion Usin the Power – law Model. Tema Tend .Mat.Apl.Comput, PP 333-340, (2003).

[5] Van Tuan Nguyen et all. “Etude Numérique de l’écoulement de fluides Non Newtoniens à travers un Elargissement Brusque », C.R.A cad .Sci Paris. T327, Série IIb, pp: 91-94. (1999).

[6] M. Mahfoud . “Frottement et Pertes de charges des fluides Visco-élastiques dans des conduites Rectangulaires. Int . Comm. Heat .Mass. Transfert. Vol 33.PP 580-590.(2006).

[7] E. Mitsoulis, J. Vlachopoulos, Non isothermal flow through parallel plates and a sudden contraction, The Can. J.Chem.Engin., 62, 837 (1984).

[8] A. Laval, Mixed convection heat transfer to power law fluid in arbitrary cross section ducts, J.Heat Transfer, 111,396 (1989).

How to cite this paper:

Belatrache D, Harrouz A, Abderrahmane A, Manaa S. Numerical Simulation of a Pseudo Plastic Fluid Through Sudden Enlargement. Algerian Journal of Renewable Energy and Sustainable Development, 2019, 1(1),92-98.

https://doi.org/10.46657/ajresd.2019.1.1.9