COB-2019-0993
NUMERICAL SIMULATION OF A PROJECTILE-FLUID IMPACT USING EULERIAN-LAGRANGIAN METHOD

Samuel Borges Costa
Antonio José Oliveira Cabral
Universidade Federal Fluminense – Escola de Engenharia Industrial Metalúrgica de Volta Redonda EEIMVR
samuelborgescosta@gmail.com
ajoc@metal.eeimvr.uff.br

Rogério Arving Serra
UniFOA - Centro Universitário de Volta Redonda
arving4@yahoo.com.br

Abstract. This paper has the objective of developing a numerical simulation of a solid-fluid interaction using the commercial software Abaqus®. A Coupled Eulerian-Lagrangian (CEL) analysis will be used to understand the entrance of a solid object in a fluid surface. The 3D model will include a chamfered cylinder of lead (projectile) with initial velocity and a static rectangular fluid body. From this paper, relevant information and steps of CEL analysis will be learned, as defining material parameters, creating appropriate mesh, applying initial condition, fluid-structure interaction and, finally, extracting data from the software. The main goal of this work is the development of a low cost ballistic projectile recovery chamber project. The authors intend to use the density and viscosity capability of some fluids to stop ballistic projectile without damaging it. Therefore, understanding the impact between the solid and the fluid and the influence of density and viscosity in this process is primordial. In this final paper is expected to see some difference in the depth penetration of the projectile when the density and viscosity of the fluid is changed. It is expected that increasing density and viscosity values will reduce the penetration depth of the projectile and also increase how fast the projectile reduce its velocity.

Keywords: Ballistic impact, projectile recovery chamber, viscosity, simulation, Coupled Eulerian-(Lagrangian) (CEL)

1. INTRODUCTION

A numerical simulation of a solid-fluid impact is the main objective of this paper. The complexity of this kind of analysis has raising debates and studies among researchers for centuries. Ahmadzadeh et al. (2014) performed simulations of a sphere entering in water; they simulated a real experiment and found that the numerical model had similar and satisfactory results. That research (Ahmadzadeh et al., 2014) studied the shape cavity, the pinch-off time and the depth of some rigid spheres with different densities falling in water by using the Coupled Eulerian-(Lagrangian) (CEL) formulation. They assumed the water as a viscous and compressible fluid. Ahmadzadeh et al. (2014) used the Aristoff (2010) work to prove the simulation developed.

The principle of the Eulerian analysis is based on the Volume of Fluid method (VOF), method that is established on a fraction function, C. When the fluid moves into the cell of the Eulerian mesh, the scalar variable C increases inside each one. If there's no fluid inside the cell, C is zero, if the cell is full, C = 1, according to Constantinescu et al. (2009).

The investigation of this paper introduces a study for a larger project, where the authors seek to develop a low cost ballistic projectile recovery chamber making use of the density and viscosity of different fluids. The necessity of recover ballistic projectiles is really important nowadays, where the use of fire guns in crimes is increasing exponentially.

The velocity, the displacement and the viscous energy behave of the projectile are some data that can be extracted from the software Abaqus®. Silveira (2013) developed a simple model for a projectile stopping in water; this paper uses that model to compare and prove the results from the numerical simulation.

The present simulation will use the CEL method. The model starts with the projectile touching the fluid surface and finish after 0.015seconds. The material properties applied will be pure lead for the projectile and water for the fluid in order to predict the impact between the solid and the fluid. The authors will check the relative error between the results from the simulation and from Abaqus®

After proving the model, it is possible to see difference between the simulation results for different fluids, displacement and velocity will be compared in line graphs.

2. NUMERICAL MODELING

According to Constantinescu et al. (2009) it is possible to use the linear Us-Up form of the Mie-Gruneisen equation of state to develop a flow modeling of a compressible fluid. This equation is linear in energy and the most common form is found in Eq. (1).

$$P - P_H = \Gamma \rho \ (E_m - E_H) \tag{1}$$

Where P is the pressure stress defined as positive in compression, E_m the internal energy per unit mass and ρ is the density of the fluid; P_H is the Hugoniot pressure and E_H is the specific Hugoniot energy. Hugoniot curve is a unique relationship between P and $1/\rho$, which is unique to the material described by the equation of state model. This curve is the locus of $P - V$ states achievable behind the shock. V is the current volume. Γ is the Grüneisen ratio given by Eq. (2), Γ_0 is a material constant and ρ_0 is the reference density.

$$\Gamma = \Gamma_0 \, \rho_0 / \rho \tag{2}$$

The linear relation U_s-U_p can be found in Eq. (3), where c_s is the sound velocity at small nominal strains and s defines this relationship. U_s is the shock velocity and U_p the particle velocity.

$$Us = cs + sUp \tag{3}$$

The linear $U_s - U_p$ equation of state model can be used to model incompressible viscous and inviscid (viscosity equal to zero) laminar flow governed by the Navier–Stokes equation. In this approach as Ahmadzadeh et al. (2014) & Constantinescu et al. (2009) applied, the equations of state govern the volumetric response, and the bulk modulus perform as an incompressible penalty parameter, $\rho_0 c_0{}^2$ is the equivalent to the elastic bulk modulus at small nominal strains.

2.1 Coupled Eulerian – (Lagrangian) (CEL) method

For Eulerian analysis, the nodes of the model are fixed in the space, and the material flows through elements that do not deform. For this approach the elements of the Eulerian domain can be full of material, partially or completely void. This kind of analysis allows effective modeling of applications involving extreme deformation, including fluid flow.

The Eulerian material boundary must be computed during each time increment. If any Eulerian material moves outside the Eulerian mesh, it will be lost from the simulation (Ahmadzadeh et al., 2014).

The Eulerian simulation potential can be associated with the Lagrangian capability, thereby allowing models between relatively rigid bodies and highly deformable materials. This kind of contact is referred as Coupled Eulerian– (Lagrangian) (CEL) analyses (Constantinescu et al., 2009).

The simulation developed in this paper also used concepts learned in Abaqus 6.10 Online Documentation (Dassault Systèmes©, 2010).

2.2 Parts and Material Properties of the model

For the impact simulation a solid chamfered cylinder with 0.01 m diameter and 0.014 m long was used, it is showed in Fig. 1 and the properties applied to it is showed in Tab. 1. The Eulerian domain is modeled as a rectangular chamber with 0.4 m x 0.4 m face and 3 m depth. The dummy part filled with the fluid was 0.01 m x 0.01 m and 2.7 m depth. The Eulerian control volume can be seen on Fig. 2.
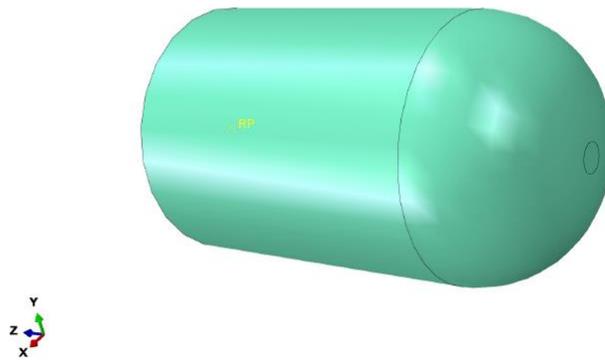
Printed using Abaqus/CAE on: Mon May 27 21:27:35 E. South America Standard Time 2019   ƷS SIMULIA

Figure 1 - Projectile part

Table 1. Properties applied to the projectile.

Properties	Lead
Density (kg/m³)	7860
Young's Modulus (Pa)	200E9
Poison ratio	0.445

Printed using Abaqus/CAE on: Sun May 26 09:23:30 E. South America Standard Time 2019   ƷS SIMULIA
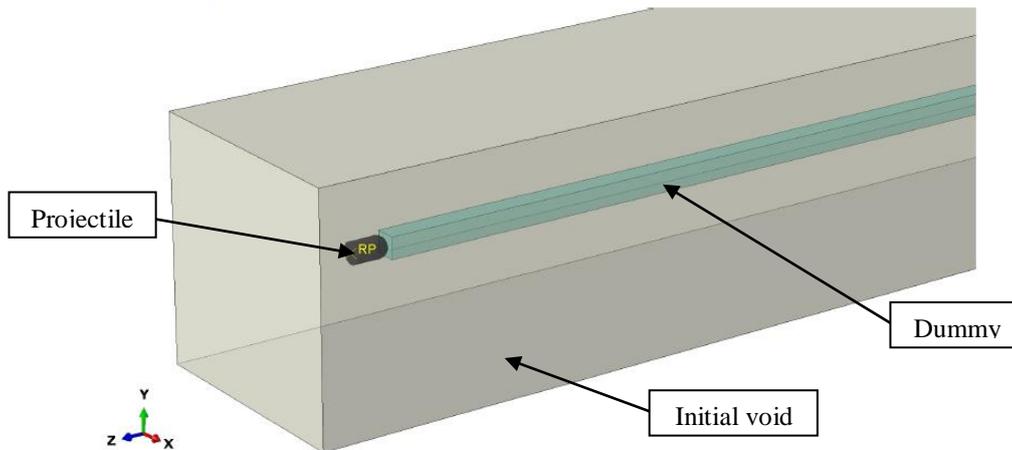
Figure 2 – Initial void region, dummy and projectile.

For the fluid, the parameters used are reported in Tab. 2. As showed before, for this approach will be used a Mie–Grüneisen equation of state (EOS) with a linear relation U_s-U_p Hugoniot form. The authors of this paper used different density and viscosity values for the fluid.

Table 2. Properties applied to the projectile.

Properties	Water	Fluid 1	Fluid 2	Fluid 3	Fluid 4
Density (kg/m³)	1000	1000	1250	1500	1750
c_0	1450	1450	1450	1450	1450
s	0	0	0	0	0
Gamma0 ($\Gamma0$)	0	0	0	0	0
Dynamic Viscosity (Pa.s)	0.001	20	30	40	50

In this paper the authors changed the density and the dynamic viscosity of the fluid in order to understand the fluid behavior. Analyzing the graph of the variation we can check that the viscosity and density has influence during the penetration of the projectile in the fluid. The values of c0, s and Γ0 are kept the same for all fluids.

2.3   Mesh, Interaction and Loads

This numerical simulation of solid-fluid contact uses explicit elements, for the Eulerian part was set ~45600 nodes with approximately 0.005 global sizes. The dummy has ~18162 nodes and 0.001 global sizes. For the solid deformable projectile were used ~310 nodes with approximately 0.005 global sizes. Figure 3 shows the mesh applied.

In order to fill the dummy part with the fluid, this part was built as a discrete rigid solid type and shaped as a shell. The Volume Fraction Tool and later the Material Assignment Load was used to fill the dummy with water.

The load applied in the projectile was initial velocities of 290 m/s. Wall boundary conditions were applied on the sides and the bottom of the Eulerian domain.

The node 196 results were chosen to represent the projectile. The values used to generate the graphs in section 4 are results of this unique nodal. This node is located on the top of the projectile.
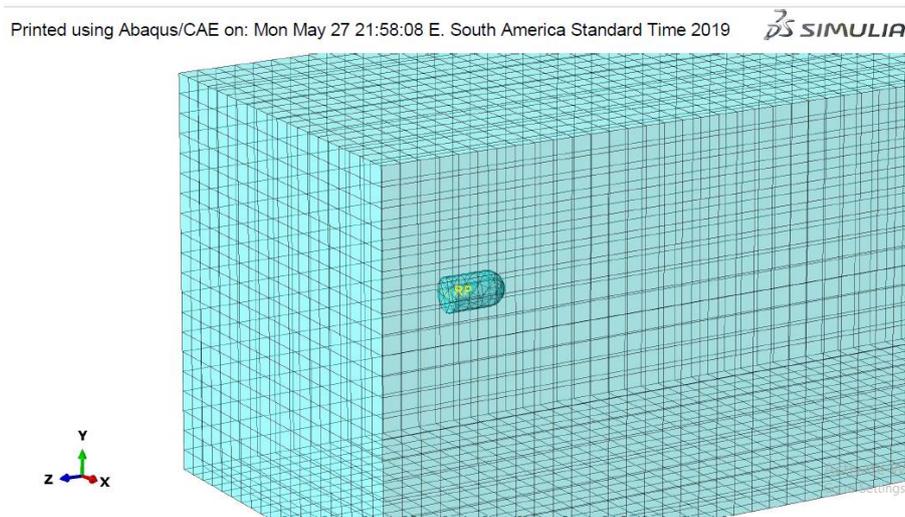

Figure 1 - Mesh

3. MATHEMATICAL MODEL

In the paper of Silveira (2013), the fluid resistance is dominantly inertial and the Initial Drag Force (F_{drag}) is applied on the projectile in an opposite direction of the velocity. The F_{drag} is function of the Drag Coefficient (C_a), section area (A) and velocity (v) of the projectile, and it is calculate in Eq. (4).

$$F_{drag} = 1/2 * C_a * \rho * A * v^2 \tag{4}$$

Velocity and displacement (x) also can be calculated. Both uses a variable k calculated in Eq. (5), which is variable of the liquid density (ρ), projectile diameter (D) and projectile mass ($m_{projetile}$).

$$k = (\rho * \pi * D^2) / (16 * m_{projetile}) \tag{5}$$

The velocity of the projectile versus time (t) is showed in Eq. (6). It is also variable of initial velocity (v_0) and k, as Eq. (7).

$$v = [k * t + v_0^{-1}]^{-1} \tag{6}$$

The displacement can be calculated by the Eq. (7).

$$x = [ln (k*t*v_0+1)] / k^1 \tag{7}$$

This article uses this model (Silveira, 2013), to prove the results of the Abaqus® simulation. The model developed using the software is showed in section 2.

4. RESULTS

It is possible to visualize the position of the projectile and how the fluid behaves after the impact. The time selected to take this picture was 0.003 sec. The values in Fig. 4 show that the penetration depth changes according to the density and viscosity variation. The fluid behaves are showed in Fig. 5, from this we understand the movement of the fluid after the impact. In the real case this displaced fluid would move the fluid around it, resulting in a resultant force on the ballistic camera wall.
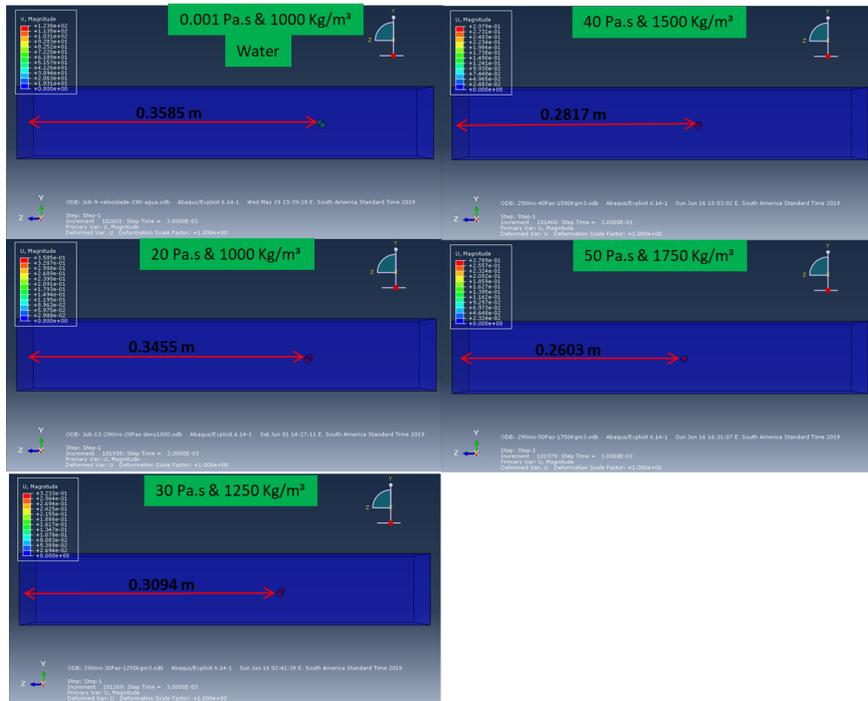

Figure 4 – Position of the projectile in 0.003 sec. / Abaqus®/ for different fluids / v_0 = 290 m/s
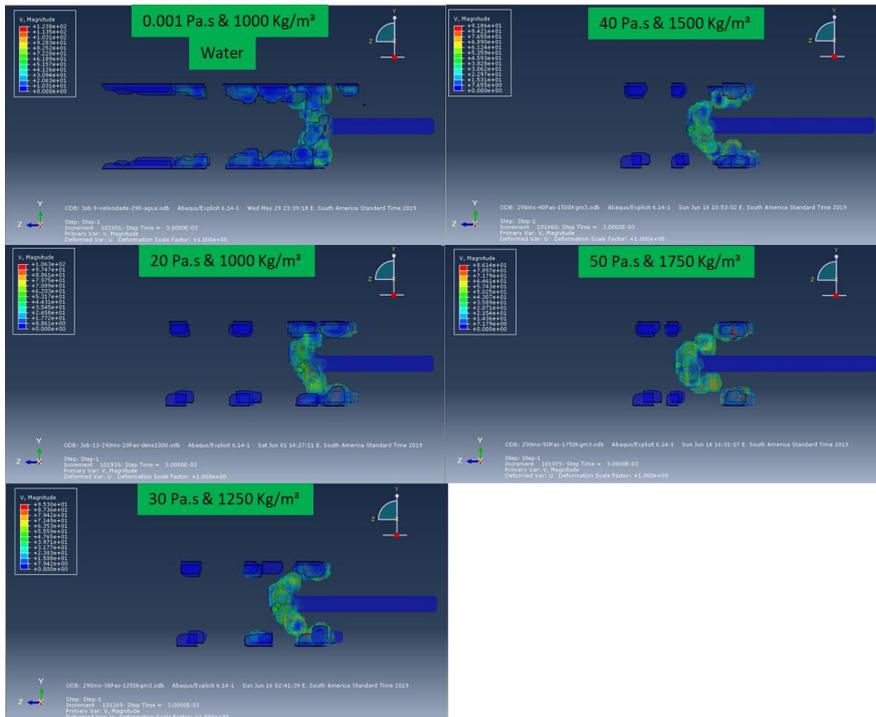

Figure 5 – Fluid behaves after 0.003 sec. / Abaqus®/ for different fluids / v_0 = 290 m/s

4.1 Validation of the simulation

In order to prove the results from the Abaqus® simulation, this paper used Silveira (2013) model. The displacement and velocity curves for the water fluid and lead projectile with 290 m/s initial velocity are showed in Fig. 6 and Fig. 7. We can check that the behaviors of the curves are similar, in spite of the difference between the results values; this difference (relative error %) is showed in table 3 and table 4.
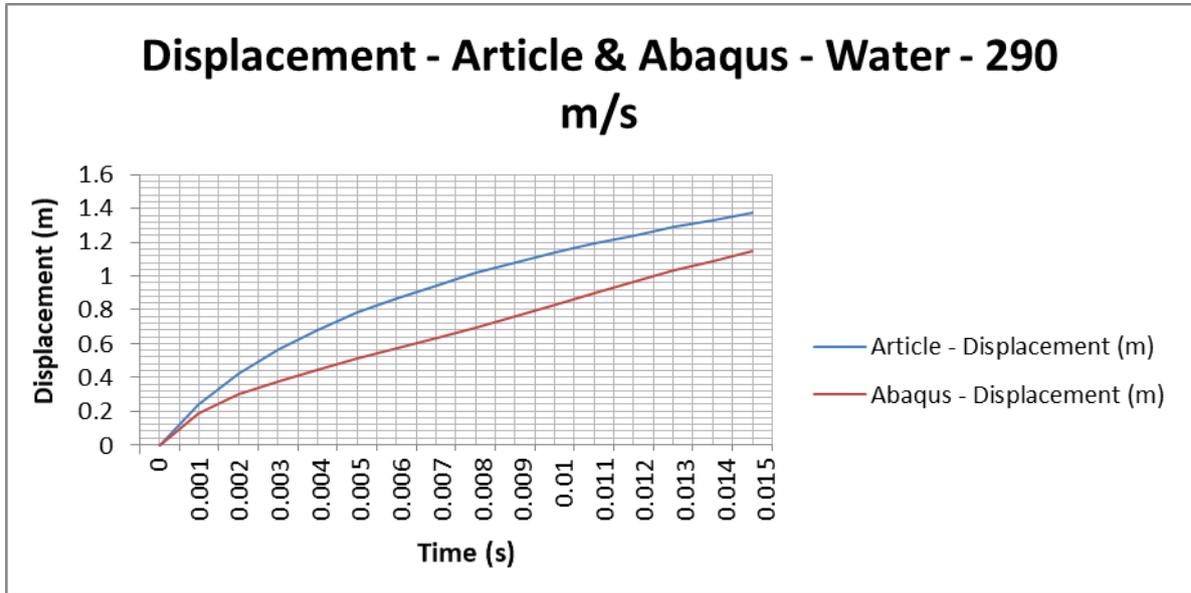

Figure 6 – Displacement curve / Article & Abaqus®/ Water / v_0 = 290 m/s

Table 3. Data to generate the displacement curve and its relative error (%).

| Time (s)	Article - Displacement (m)	Abaqus® - Displacement (m)	Relative error (%)
0	0	0	
0.001	0.242183805	0.18652418	23%
0.002	0.421206185	0.297724429	29%
0.003	0.563288812	0.373036019	34%
0.004	0.681099063	0.44439341	35%
0.005	0.781732441	0.512752575	34%
0.006	0.869566268	0.576017133	34%
0.007	0.947492152	0.635048182	33%
0.008	1.017520515	0.695022299	32%
0.009	1.08110585	0.759328073	30%
0.01	1.139334501	0.829441218	27%
0.011	1.193039298	0.900902034	24%
0.012	1.242872776	0.969672803	22%
0.013	1.289355733	1.032366691	20%
0.014	1.332910464	1.091559935	18%
0.015	1.373884139	1.151348829	16%
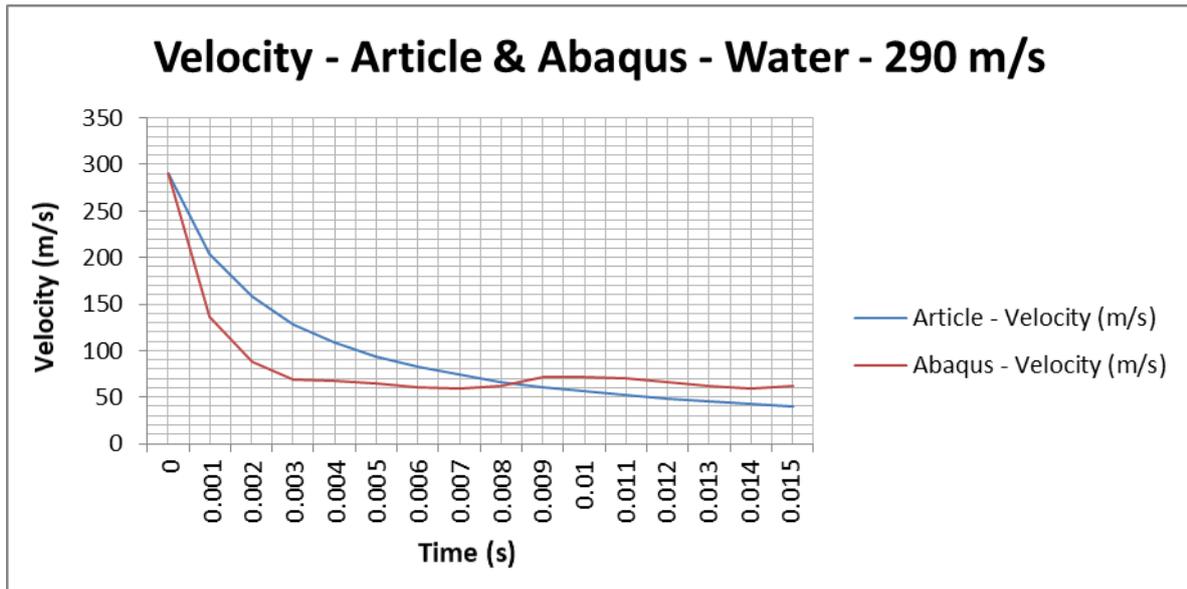
Figure 7 – Velocity curve / Article & Abaqus® / Water / v_0 = 290 m/s

Table 4. Data to generate the velocity curve and relative error (%).

Tempo (s)	Article - Velocity (m/s)	Abaqus® - Velocity (m/s)	Relative error (%)
0	290	290	
0.001	204.3267213	136.1636667	33%
0.002	157.7294754	88.46184487	44%
0.003	128.4386937	69.30476251	46%
0.004	108.3228527	67.03718654	38%
0.005	93.6548032	64.54672717	31%
0.006	82.48541369	60.07269564	27%
0.007	73.69630032	58.53162032	21%
0.008	66.59985008	62.54438673	6%
0.009	60.7500366	71.74912861	18%
0.01	55.84488776	72.02253923	29%
0.011	51.67267372	69.69707695	35%
0.012	48.08054001	65.85014655	37%
0.013	44.95537343	61.30219735	36%
0.014	42.21167474	59.77112497	42%
0.015	39.78361668	61.78344345	55%

4.2   Density and viscosity variation in the simulation

After proving the results, and making sure the curves of the simulation and the article (Silveira, 2013) have the same behavior. The authors changed the density and viscosity of the fluid for the same projectile (material and initial velocity). The displacement and velocity rate reduce changed for each simulation. The different results can be seen on Fig. 8 and Fig. 9. It is observed that for the fluid with 30 Pa.s and 1250 Kg/m³ the penetration shows a troubled behavior.
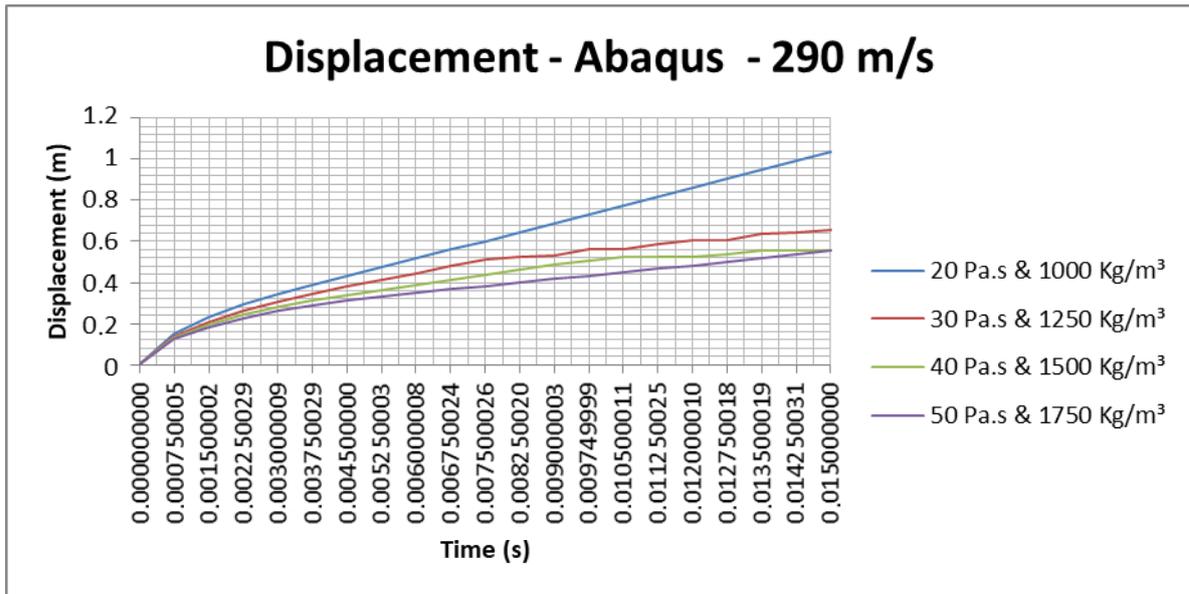
Figure 8 – Displacement curve / Abaqus® / for different fluids / v_0 = 290 m/s
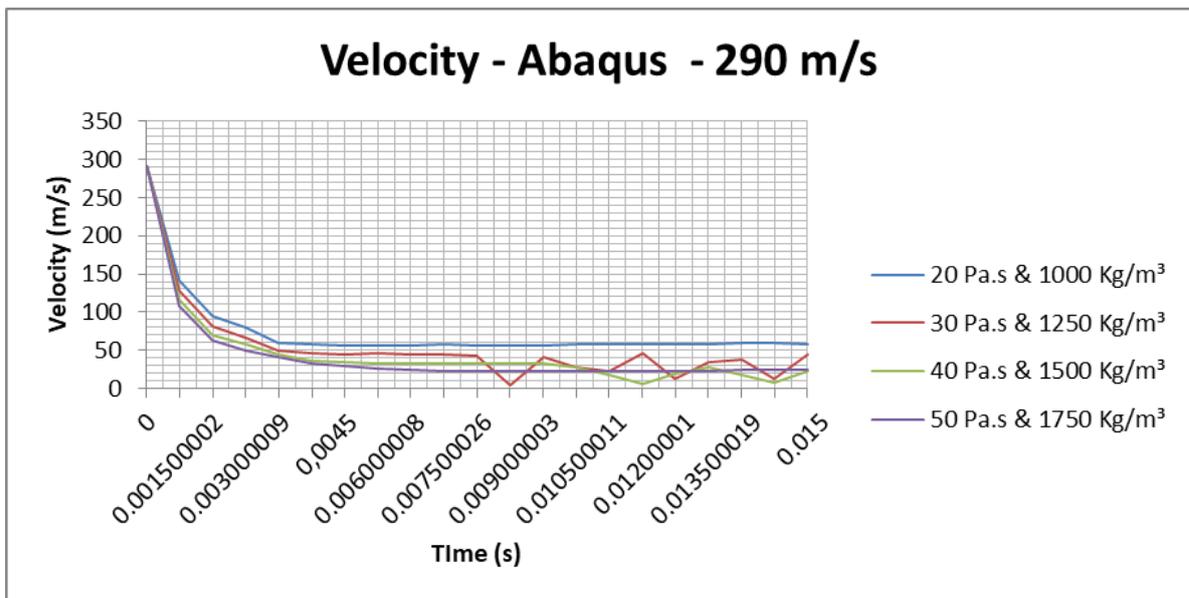

Figure 9 – Velocity curve / Abaqus® /for different fluids / v_0 = 290 m/s

4.3   Reynolds Number for the initial velocity of the simulations

Reynolds number can be calculated for water and for each fluid simulated in Abaqus®. This property is compared in Fig. (8). The initial velocity and the body dimension were fixed; density and viscosity were changed.

It is possible to verify a laminar behavior on the fluids (1, 2, 3 and 4) used in the simulation (142 < Reynolds number < 203). This is different when the Reynolds number is calculated for water, in this case (water) the behavior of the system is turbulent (Reynolds number ~4060000). The comparison graph in Fig. 8 uses a logarithmic scale for the Reynolds number.

Reynold Number for different fluids - 290 m/s
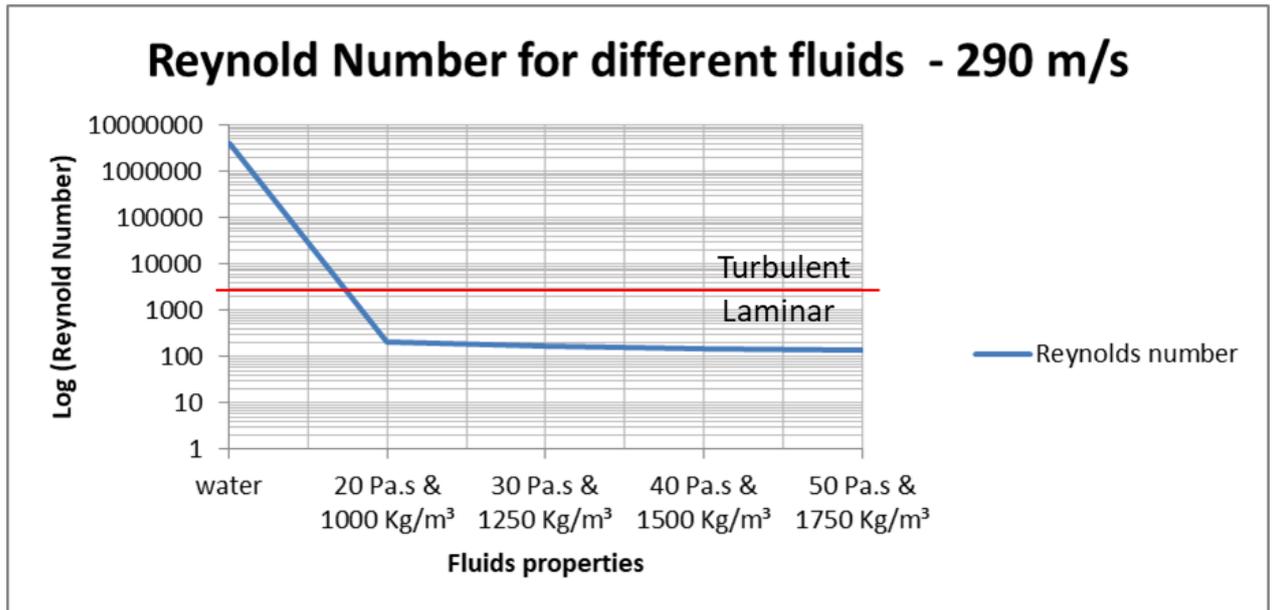
5. CONCLUSION

As expected, this paper results showed that the simulation developed was a good prediction of the projectile behavior after the impact with the fluid. This paper also proved that the density and viscosity of the fluid have influence in the projectile behave after the impact; the bigger density and viscosity, the bigger the rate of reduce velocity and less the displacement of the fluid.

In conclusion the authors of this paper can use the model developed to choose the ideal properties of the fluid to stop a ballistic projectile in a short distance.

The authors propose a future work with an analysis of the stress and strains of the projectile, however, the properties applied to the projectile should be the same as the real ballistic projectile.

6. ACKNOWLEDGEMENTS

We thank PPGEM from Universidade Federal Fluminense (EEIMVR) and the Polícia Civil do Rio de Janeiro for the partnership on this paper. We hope that the development of this work will be useful.

7. REFERENCES

Ahmadzadeh, M., Saranjama, B., Hoseini Farda, A., Bineshb, A.R., 2014. "Numerical simulation of sphere water entry problem using Eulerian–Lagrangian method". *Applied Mathematical Modelling*, Vol. 38, pp.1673-1684.

Aristoff, M., Truscott, T. T., Techet, A., Bush, J. W., 2010. "The water entry of decelerating spheres". *Physics of Fluids - 22. 10.1063/1.3309454*, Vol. 22.

Constantinescu, A., Alaoui, A. E. M., Neme, A., Rigo, P., 2009. "Numerical and Experimental Investigations of Simple 2-D Geometries in Slamming". *In 10th Inter Marine Design Conference 2009*, Trondheim, Norway.

Dassault Systèmes©, 2010. "Eulerian analysis". *Abaqus 6.10 Online Documentation*. 28 Apr. 2010 <https://www.sharcnet.ca/Software/Abaqus610/Documentation/docs/v6.10/books/usb/default.htm?startat=pt04ch13 s01aus85.html>.

Silveira, F. L., 2013. Frenagem de um projétil em um meio fluido: "Qual seria a distância, dentro da água, percorrida por um projétil calibre .50 com massa de 50g e velocidade de 850 m/s?". *Caderno Brasileiro de Ensino de Física*, Vol. 30, pp. 156-164.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.