The advance experimental and numerical studies of water turbulent jets

Dan Victor CAVAROPOL

Abstract: this article presents recent research in the field of free shear flows, especially those of the turbulent jets. The intervention in the extinguishing process in the real fire needs more information about the zones, and the splitting process, the diameters of the drops formed. The natural scale experiments were carried out, and also a numerical simulation, considering the instability of a water turbulent jet. These results give important information about how and where must be a firefighter positioned in order to extinguish a fire in a shorter time, with a minimum amount of water.

Key-Words: Fire extinguishing, Turbulence, Water jets, Intervention, Modeling

1. The turbulent water jets used in fire extinguishing process

The liquid jet of water is a turbulent water jet and example of a free shear flow, and is very useful for any kind of intervention in case of a fire.
The construction of a new nozzle for the water sprays needs also many studies, numerical and experimental ones.
We utilise the package Fluent/Gambit (finite volume method) to simulate the flow of water inside an elipsocyclonic nozzle. In defining the geometry of the model, we have imported in the Gambit module the solid model (Fig.1), made with the SOLID WORKS and saved in IGES form.
The space is discretised in a mixt structure of the tetraedrals, with a number of 580,000 volumes.

Author is with the Academia de Politie “Alexandru Ioan Cuza”, Dean of the Facultatea de Pompieri, București. Str.Morarilor, nr.3, sector 2, București, România (corresponding author to provide phone: +40021.255.56.68; fax: +40021.255.56.63; e-mail: cavaropold@gmail.com).

If we consider the incompressibility of the fluid (water), and neglecting the external forces, the equations of the flow are, in cylindrical coordinates (r, φ, z):

\[
\frac{dw}{d\tau} = \frac{w_i^2}{r} = -\frac{1}{\rho} \frac{\partial p}{\partial r}, \quad (1)
\]

\[
\frac{dw}{d\tau} + \frac{w_i}{r} \frac{dr}{d\tau} = -\frac{1}{\rho r} \frac{\partial p}{\partial \phi}, \quad (2)
\]

\[
\frac{dw}{d\tau} = \frac{1}{\rho} \frac{\partial p}{\partial z}
\]
In the above equations, the meanings of the terms are such as:

- $r$-radial direction,
- $\tau$-time,
- $\rho$-density of the fluid,
- $p$-pressure,
- $w$-speed of fluid.

The components of the speed are tangential $w_t$, and radial $w_r$.

The rotating flow inside the nozzle is characterized by the condition:

$$w_r r^2 = \omega$$

and the flow rate $Q_i$:

$$Q_i = r_i^2 \pi w_i = 4 r_i r_w$$

(4)

A steady internal and axisymetric flow is:

$$\frac{dw_r}{d\tau} = 0, \frac{dw_t}{d\tau} = 0$$

(5)

the variation of the speed in time are:

$$\frac{d\omega_t}{d\tau} = \frac{\partial \omega_t}{\partial \tau} w_t + \frac{\partial \omega_t}{\partial r} w_r + \frac{\partial \omega_t}{\partial z} w_z = 0$$

(6)

$$\frac{d\omega_r}{d\tau} = \frac{\partial \omega_r}{\partial \tau} w_t + \frac{\partial \omega_r}{\partial r} w_r + \frac{\partial \omega_r}{\partial z} w_z = 0$$

(7)

The variation of the pressure is:

$$\frac{dp}{dr} = \frac{\partial p}{\partial r} = \rho \left( \frac{Q}{4\pi r^2} \right) \frac{1}{r^3}$$

(8)

In the nozzle the flow is turbulent, and we have adopted THE Reynolds Stress Model

The input speed of water in the nozzle considered was $V_i = 10 \text{ m/s}$. For the interior flow fields the parameters of water were introduced and for the boundary the aluminium parameters.

For the pressure, the most recommended scheme is PRESTO (PREssure STaggering Option). The results of the trajectories of the particles inside the nozzle, after 1000 iterations is depicted in figures 6 and 7.

It is important to observe that, for a final exit of diameter $d=12 \text{ mm}$, the speed is four times greater $(40 \text{ m/s})$ than the initial speed $(10 \text{ m/s})$.

In general, the water jets are turbulent, with Reynolds $Re>1200$ at the beginning of the first zone. In the first part-the compact jet – the speed is very high and it can be used for destroying the basement of the fire (at the ground level of the fire).

The force of the jet can be calculated such as:

$$R_i = 1.57 d_i^2 p$$

(9)

$d_i$ being the diameter of the jet and $p$ is the pressure. If the pressure is bigger than $5...6$ bars, the pipe must be handled by two firefighters.

In the second (transition) zone, the air begins to enter in the jet, creating many air bubbles.

In the third zone, the drops zone (with diameters between $0.01...5 \text{ mm}$), we have an exchange of heat, mass and momentum between the jet and the surrounding area. The
optimal drops diameter for firefighting is in the range 0.1...1 mm, the others drops (diameters between 0.01...0.04 mm) creating a fog. The total length of the jet is a sum of the lengths of three zones.

The total (maximum) horizontal length of a jet is:

\[ L_{\text{max}} = \delta H_v \]  
(10)

where \( H_v \) is the vertical height of a jet with the same characteristics, \( \delta \) is an experimental number, depending of the angle of the pipe with orizontal plane.

We can calculate the flow rate of a water jet with the formula:

\[ Q = \mu A \sqrt{2gh} \]  
(11)

where:

- \( A \) - initial area of the jet,
- \( \mu = 0.622 \) for round jets

(like in our experimental results)

\( h \) - the high of the same vertical water round jet.

But \( \mu \) is depending of the Reynolds and Weber:

\[ \mu = f(Re,We) = f\left( \frac{\nu d_0}{\rho g h d_0} \right) \]  
(12)

For liquid round jets, in cylindrical coordinates, using the \( k-\varepsilon \) model, the equation used for the dissipation \( \varepsilon \) is:

\[ u_c \frac{\partial u}{\partial x} + u_r \frac{\partial u}{\partial r} = c_{\varepsilon_k} \frac{\partial}{\partial x} \left( \frac{\varepsilon}{k} \right) + c_{\varepsilon_r} \frac{\partial}{\partial r} \left( \frac{\varepsilon}{r} \right) \]  
(13)

where:

- \( k \) - kinetic energy;
- \( x, r \) - coordinates;
- \( P \) - production of energy;
- \( C_{\varepsilon} = 0.09 \)
- \( c_{\varepsilon_k} = 1.44 \)

2. Descriptions of the jets movement

In simplified maner, we can use the free shear flow of a bubble of water with the diameter equal with the initial water jet.

The friction force can be written like in the following equations:

\[ \vec{R} = -mc_v \vec{v} \]  
(14)

where:

- \( m \) - the weight of the water sphere;
- \( \vec{v} \) - the speed in direction Ox;
- \( c_v \) - the friction factor.

In scalar coordinates:

\[ m \frac{d^2 x}{dt^2} = -mc_x \frac{dx}{dt} \]  
(15)

\[ m \frac{d^2 y}{dt^2} = -mg - mc_y \frac{dy}{dt} \]

After calculation the equation of \( y=f(x) \) is

\[ y = x \tan(\alpha) + \frac{gx}{c_v v_0 \cos(\alpha)} - \frac{g}{c_x} \ln \frac{v_0 \cos(\alpha)}{v_0 \cos(\alpha) - c_v x} \]  
(16)

3 Experimental results

The experiments were made at natural scale, utilising the romanian engin for firefighting Roman As.P.Ls R 8135 (firefighting engins for water and foam).

The 100 collectors of the water jets (arranged in a matrix of 10x10), with the following dimensions 200*200*100 mm (total surface of 0.04 m\(^2\)) are made of iron. The jet is supposed to be round and symmetric at the beginning, and have enough energy (momentum) to reach the fire. Of course, the final part of the jet is in fact made water drops with different diameters, between 1 mm and 0.1 mm.

We present some of the mains experiments, showing the spread of the round turbulent jet and the total amount of water collected in each iron box.

The experimental results confirm the theoretical ecuations results, as is presented in the following tables.

In the first examples, the pressure remain unchanged and the diameters and angle of the water jet are changed. In the last example, it is changed the jet pressure also.

First experiment – initial values of the parameters (pressure, diameter, angle and time)-see Table No.1

- pressure-3 bars;
- starting diameter of turbulent jet -Φ 14;
- angle of water jet-30°;
- collecting time - 60 s.

Second experiment – initial values of the parameters (pressure, diameter, angle and time)-see Table No.2

- pressure-3 bars;
- starting diameter of turbulent jet -Φ 12;
- angle of water jet-23°;

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
<th>H</th>
<th>I</th>
<th>J</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>1.2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>0.1</td>
<td>1.8</td>
<td>3.2</td>
<td>1.7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table no.1
4 Conclusion

The model presented is developed considering the fluid parameters as for pure water. Following the presented results, the theoretic model is sufficient similar to the experimental results.

The model can be improved by considering new fluid parameters, taking into consideration the actual additive substances added to the fire jets. The main changed parameters are viscosity, density and capillary coefficient. Another parameter to be considered is the meteoric influence on the jet aspect, especially the wind direction and speed.

The jet efficiency is a practical topic, considering both the necessity of the fire devices effectiveness and also the actual energy saving presents the interest for fire devices and equipments.

Considering the scientific literature consulted, the new models are guiding static and dynamic devices.

References:
