NUMERICAL MODELLING OF COMBUSTION PROCESS WITH THE USE OF ANSYS FLUENT CODE

Miroslaw Kowalski
Air Force Institute of Technology
Ksiecia Boleslawa 6, 01-494 Warsaw, Poland
tel.: +48 22 851301, fax: +48 22 851313
e-mail: miroslaw.kowalski@itwl.pl

Antoni Jankowski
Institute of Aviation
Krakowska Avenue 110/114, 02-256 Warsaw, Poland
tel.: +48 22 8460011, fax: +48 22 846432
e-mail: antoni.jankowski@ilot.edu.pl

Abstract

The article presents the modelling of the combustion process of liquid fuels using professional ANSYS FLUENT software. This program allows modelling the dynamics of compressible and incompressible, laminar and turbulent flows as well as heat exchange phenomena with occurrence and without chemical reactions. The model presented in the article takes into account the influence of the gas phase on the liquid phase during the fuel combustion process. The influence of velocity and pressure of the flowing gas and the type of flow has a significant impact on the combustion of liquid fuels. The developed model is fully reliable and the presented results are consistent with experimental research. The occurrence of a laminar sublayer in a turbulent flow was confirmed, and the thickness of this layer and the turbulent layer significantly influences the course of the combustion process. The use of the flat flow model reflects the basic phenomena occurring during the combustion of liquid fuels under turbulent conditions. The use of the program for flows with different flow velocity profiles is justified. It gives important information about the processes taking place during the combustion of liquid fuels. The results of numerical tests are presented graphically. The article presents graphs of velocity field, absolute pressure, power lines, temperature and density.

Keywords: liquid fuels, combustion processes, turbulent flow, combustion process model, ANSYS FLUENT

1 Introduction

In order to compare the experimental results obtained in tests on a special position in which the flow was realized with the addition of the weight, with the capabilities of a theoretical analysis of such phenomena occurring in the flow, calculations based on the modelling of the flow in the pipe were performed. In modelling, the ANSYS FLUENT program, which gives the possibility to model a wide range of issues related to computational flow dynamics (CFD), both in relation to the flow of compressible and incompressible fluids, laminar and turbulent flows, phenomena of transport, heat exchange, occurrence and without occurrence chemical reactions etc. was applied. The program allows analysing steady, transient and unsteady flows. Particular emphasis in this program is on the modelling of the phenomena occurring during combustion, which includes models of energy dissipation and models of the probability density function. If there is heat exchange or compressibility in the flows, the energy conservation equation is also solved. With regard to flows, in which substance mixing occurs, chemical reactions occur, particle conservation equation, or in the case of the non-premixed combustion model, the equations of the conservation of the mixture fractions and their variances are solved. Turbulent flows are characterized by fluctuations in velocity fields. These fluctuations cause the mixing of the transport size, such as moment, energy, particle
concentrations, and are the reason for the fluctuation of transported quantities. Because these fluctuations can occur on a small scale, but with high frequency, they are time-consuming in engineering practice for direct calculations. Therefore, instead of these equations, which relate to a given moment, the equations averaged over a period may be given, or they may be modified in such a way as to remove solutions relating to a small scale, which greatly simplifies the solution of equations by computer methods. However, the modified equations may contain additional unknown variables, and therefore it was necessary to introduce turbulence models allowing determining these variables in relation to known models. There are many models of turbulence, the use of which depends on the course of the phenomenon being studied.

2 Results of numerical calculations

The calculation domain scheme

The calculations concerned turbulent flow, in which the mass of the factor was added at the bottom of the domain (Figure 1). The upper and sidewalls were set as isothermal with a temperature of 300 K. Additionally, the slip option was chosen for them. For the $V_d$ 0.05 and 0.1 m s$^{-1}$ velocities, the laminar flow model was used, and for 1 m s$^{-1}$ velocity – the SST turbulent flow model, which was developed for solutions of similar cases.

![Fig. 1. Diagram of a two-dimensional computational domain with the addition of mass at the bottom of the channel for $T = 300$ K](image)

The parameters of the analysed cases are presented in Tab. 1. The calculations for 9 variants were made. Velocity $V_d$, and $T_d$ temperature side stream added.

Tab. 1. 9 variants of calculations

<table>
<thead>
<tr>
<th>Item</th>
<th>$V_d$</th>
<th>$T_d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.05 m/s</td>
<td>300 K</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>500 K</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>1 000 K</td>
</tr>
<tr>
<td>4</td>
<td>0.1 m/s</td>
<td>300 K</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>500 K</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>1 000 K</td>
</tr>
<tr>
<td>7</td>
<td>1.0 m/s</td>
<td>300 K</td>
</tr>
<tr>
<td>8</td>
<td></td>
<td>500 K</td>
</tr>
<tr>
<td>9</td>
<td></td>
<td>1 000 K</td>
</tr>
</tbody>
</table>

Variant 1

Fig. from 1.1 to Fig. 1.6 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity composition $V_y$, absolute pressure $p$ and vector field, vector field of velocity and stream lines.
Variant 2

Fig. from 2.1 to Fig. 2.8 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity composition $V_y$, absolute pressure $p$, vector field of velocity and stream lines, Temperature $T$ and density $\rho$. 
Numerical Modelling of Combustion Process with the Use of Ansys Fluent Code

**Fig. 2.4. Absolute pressure**

**Fig. 2.5. Vector field of velocity**

**Fig. 2.6. Stream lines**

**Fig. 2.7. Temperature field**

**Fig. 2.8. Density field**

**Variant 3**

In Fig. from 3.1 to 3.8 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity component $V_y$, absolute pressure $p$, vector field of velocity and stream lines, Temperature $T$ and density $\rho$.

**Fig. 3.1. Velocity**

**Fig. 3.2. Velocity component $V_x$**

**Fig. 3.3. Velocity component $V_y$**

**Fig. 3.4. Absolute pressure**

178
Variant 4

In Fig. from 4.1 to 4.6 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity composition $V_y$, absolute pressure $p$, vector field of velocity and stream lines.
Variant 5

In Fig. from 5.1 to 5.8 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity composition $V_y$, absolute pressure $p$, vector field of velocity and stream lines, Temperature $T$ and density $\rho$. 
Variant 6

In Fig. from 6.1 to 6.8 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity composition $V_y$, absolute pressure $p$, vector field of velocity and stream lines, Temperature $T$ and density $\rho$.

Variant 7

In Fig. from 7.1 to 7.6 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity composition $V_y$, absolute pressure $p$, vector field of velocity and stream lines, Temperature $T$ and density $\rho$. 
Variant 8

In Fig. from 8.1 to 8.8 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity composition $V_y$, absolute pressure $p$, vector field of velocity and stream lines, Temperature $T$ and density $\rho$. 
Variant 9

In Fig. from 9.1 to 9.8 show diagrams of fields: velocity $V$, velocity component $V_x$, velocity composition $V_y$, absolute pressure $p$, vector field of velocity and stream lines, Temperature $T$ and density $\rho$.  

Fig. 8.4. Absolute pressure

Fig. 8.5. Vector field of velocity

Fig. 8.6. Stream lines

Fig. 8.7. Temperature field

Fig. 8.8. Density field

Fig. 9.1. Velocity

Fig. 9.2. Velocity component $V_x$

Fig. 9.3. Velocity component $V_y$

Fig. 9.4. Absolute pressure
3. Determination of the average speed

The distance from the beginning of the channel to the place, where the end of the air inlet is located was divided into six parts, every 100 m, and then in each of the cross-sections, the average values of horizontal velocity components $V_x$ were determined. In the following diagrams (Fig. 10-12) they are presented for different velocities $V_d$ and temperature $T_d$.

Using the simulation results shown in Fig. 1-9, the relationship between the velocity component $V_x$ and the distance from the beginning of the channel for different values of temperature $T_d$ was determined. Fig. 10 shows the graph for the speed $V_d = 0.05 \text{ m/s}$, and in Fig. 69 for the speed $V_d = 0.10 \text{ m/s}$. The velocity $V_d$ models the velocity of adding a mass, which is gas with a temperature $T_d$.

In Fig. 12, the graph for velocity $V_d = 1.0 \text{ m/s}$ with reference to different values of the temperature of the added factor $T_d$ is presented. The graphs in Fig. 68-70 show that the velocity of $V_x$ increased as a function of the length of the channel and the higher the speed of adding mass $V_d$, the smallest differences in the value of the velocity component $V_x$ were observed at different temperature values. Fig. 12 shows a graph for the velocity $V_d = 1.0 \text{ m/s}$ with reference to different values of the temperature of the added factor $T_d$. The graphs in Fig. 68-70 show that the speed $V_x$ increased as a function of the length of the channel and the higher the speed of adding mass $V_d$, the smallest differences in the value of the component velocity $V_x$ were observed at different temperature values. The calculations presented model the flows in the test chamber on the research bench. The following distributions were obtained: velocity, pressure, vector field of the tide, current line, temperature field and density field in a test chamber having a rectangular cross-section. Then, the axial velocity waveforms along the length for different mass addition rates and different temperatures of the added gas medium were compared on the graphs.

4. Conclusions

The theoretical model of the combustion process relating to liquid fuels has been developed. It takes into account the influence of the gas phase, since combustion is always present in this phase, and interacts with the liquid phase. We found that the velocity gas stream and the type of flow (laminar, transitional, or turbulent) have a significant impact on the burning liquid fuels, and the impact of pressure always exists, which forces the gas stream.
Fig. 10. Average speed $V_x$ as a function of distance for $V_d = 0.05 \text{ m/s}$

Fig. 11. Average speed $V_x$ as a function of distance for $V_d = 0.1 \text{ m/s}$

Fig. 12. Average speed $V_x$ as a function of distance for $V_d = 1.00 \text{ m/s}$
Developed a theoretical model burning has a good reference for an experiment conducted in conditions of model confirmed using numerical and experimental studies of the boundary layer. The results of numerical calculations confirmed the presence of the laminar sublayer in the turbulent flow. The thickness of the sublayer laminar and turbulent flow conditions laminar under laminar flow conditions has a significant impact on the course of the combustion process. The use of a flat flow model reflects the basic phenomena that occur during combustion of liquid fuels in turbulent conditions. In terms of turbulent flows basic relationship used in the modelling using ANSYS FLUENT were presented. Calculations for flows with different flow velocity profiles, modelling add mass and different values of module velocity were carried out. Velocity $V_d$ modelling adding mass reflects the combustion process (adding mass to a stream of flowing gas). In place of breakdown the characteristics of velocity $V_d$ on the inlet is an evident increase in the velocity component $V_y$ with very small changes of this component in the rest of the channel. In the case of a fault this area increased significantly, and the breakdown velocity is greater, the greater the disturbance component of the velocity field.

References


