# Numerical simulation of fuel spraying into an industrial object

*Vojtech* Betak[1,*]

[1]Czech Aerospace Research Centre, Department of Engines, 19905 Prague, Czech Republic

**Abstract.** Numerical simulation of fuel spraying into industrial objects is described in this paper. This spatter may be caused by a vehicle or storage tank accident. Prediction of fuel spill is the main aim of simulations that allow predicting possible sources of secondary fires. Numerical simulations are based on the solution of the system of Navier-Stokes equations complemented by the solution of Lagrangian spray cloud and thin fluid film. The simulation results will help to identify better fire prevention and other preventive measures to minimize fire damage.

## 1 Introduction

In modern times, the industry is focused on various numerical simulations in the projection phase. This allows reducing the production cost, to improve operative cost and safety features. In the case of large industrial buildings, the simulation allows us to better dimension e.g. ventilation. The effect and propagation of fire can now be studied in projection phase due to increased computing power and software availability. It is especially important in the case of fire safety, where it is possible to determine safe escape routes and regulate the amount of smoke.

Results of simulating fuel spraying into an industrial building are presented in the following paper. This can be caused by a vehicle accident when the fuel tank is broken, and fuel is sprayed/spilled into the building. When the fuel spray is fine enough, it may catch fire.

The simulations are based on previous experience with liquid fuel combustion in aircraft engines [1] which are modified for large scale fire.

## 2 Mathematical model

The mathematical model of reacting fluid flow is based on the finite volume method which is applied to the extended system of Navier-Stokes equations

$$\frac{\partial}{\partial t}\rho + \frac{\partial}{\partial x_j}(\rho u_j) = S_\rho, \tag{1}$$

$$\frac{\partial}{\partial t}\rho u_i + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right)\right] + S_u, \tag{2}$$

$$\frac{\partial}{\partial t}\rho h + \frac{\partial}{\partial x_j}(\rho h u_j) = \frac{\partial}{\partial x_j}\left[\left(\alpha + \frac{\mu_t}{Pr_t}\right)\frac{\partial h}{\partial x_j}\right] + S_h, \tag{3}$$

$$\frac{\partial}{\partial t}\rho Y_i + \frac{\partial}{\partial x_j}(\rho Y_i u_j) = \frac{\partial}{\partial x_j}\left[(\mu + \mu_t)\frac{\partial Y_i}{\partial x_j}\right] + S_Y. \tag{4}$$

where density is represented by $\rho$, components of velocity vector are represented by $u_i$, pressure is represented by $p$, dynamic viscosity is represented by $\mu$. The energy equation is written in the form of enthalpy $h$ where heat diffusion is represented by $\alpha$. Turbulent quantities are represented by turbulent viscosity $\mu_t$ and turbulent Prandtl number $Pr_t$. The transport equation for fraction of species $Y_i$ must be solved represent volume fraction of species and $S_\rho, S_u, S_h, S_Y$ source term in corresponding equations.

These equations must be completed by the equation of state $p = f(\rho)$. First, we must define the molar fraction as

$$y_i = Y_i \frac{M}{M_i}. \tag{5}$$

where $M$ is an average molar mass and $M_i$ is a molar Mass of fraction i, Then is applied Dalton Law

$$p_i = \rho_i n_i R T. \tag{6}$$

where the partial pressure of fraction $i$ is defined as $p_i$, partial density is defined as $\rho_i$and number of moles of fraction i is defined as $n_i$. Temperature is represented by $T$and universal gas constant by $R$. Finally, we obtain density in the following form

$$y_i = \frac{p_i}{p}, \tag{7}$$

$$\rho = \sum y_i \rho_i \tag{8}$$

* Corresponding author: betak@vzlu.cz

and the temperature is defined as

$$T = \frac{h}{\sum y_i c_{p_i}}. \quad (9)$$

The system is complemented by Menter [2] two equations $k - \omega SST$ model which is defined as

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}[\Gamma_k \frac{\partial k}{\partial x_j}] + G_k - Y_k, \quad (10)$$

$$\frac{\partial \rho \omega}{\partial x_j} + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j}[\Gamma_\omega \frac{\partial \omega}{\partial x_j}] + G_\omega - Y_\omega + D_\omega, \quad (11)$$

where the kinetic energy of turbulence is defined as $k$ and the specific speed of dissipation as ω. This model is used in DES formulation.

The ODE system is used to describe the movement of the fuel spray particles

$$\frac{du_p}{dt} = F_D(u - u_p) + g\frac{\rho_p - \rho}{\rho_p} + F_x, \quad (12)$$

$$F_D = \frac{18}{24}\frac{\rho}{\rho_p d_p} C_D Re \quad (13)$$

and

$$F_X = \frac{\rho}{\rho_p}\left[0.5\frac{d}{dt}(u - u_p) + u_p\frac{\partial u}{\partial x}\right] \quad (14)$$

where $u_p$ is represented speed vector of spray particles and $\rho_p$ is represented fluid density of spray particles The drag coefficient $C_D = \frac{24}{Re}$ is depended as a function of modified Reynolds number $Re = \rho d_p \frac{|u - u_p|}{\mu}$.

## 3 Results

Several scenarios of fuel spray are solved in this project. These scenarios are associated with the crash of a small commercial aircraft into a building of critical infrastructure facility (for example power plants, fuel storage, chemical industry, ...). Significant environmental and economic damage is achieved in a wide area in case of accident near to these buildings.

Building geometry (Fig. 1) is selected for the demonstration of numerical simulations. Due to the impact, the sidewall is broken, and the inner space is exposed.

The simulation is initialized by approx. 12000 spray particles with initial velocity of 180 m/s and diameter 0.05 m. These droplets are moving through the computational domain where they hit the walls and break up into smaller droplets. Total pressure and pressure directed velocity boundary conditions are prescribed at the outlet from the computational domain where ambient atmosphere conditions are assumed.

It is assumed that the wall is broken at the moment of the aircraft's impact and debris and airframe disappear. At the same time, the fuel is heated by 200 degrees which initialized evaporation and burning of fuel gases.

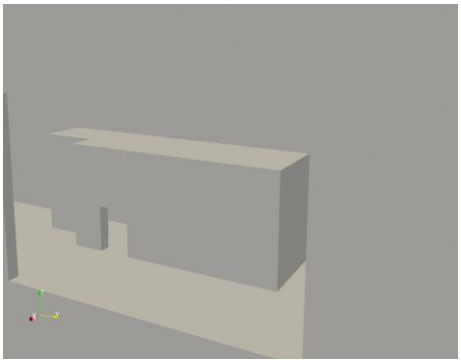

**Fig. 1.** Example of computational geometry.

Two types of simulations are solved. In the first simulation, the chemistry is switched off and therefore the drops do not evaporate and burn. In the second one, the chemistry is included.

In figures 2 and 3 is shown the difference between reacting and nonreacting simulations. In the case of reacting simulations is formed very fine fuel spray with average droplets diameter in the range from $10^{-5}$ to $10^{-4}$ m. Fuel particles are floated due to buoyancy forces caused by fire. In contrast in the case of nonreacting simulations where bigger droplets are formed. The average droplet diameter is in the range from $5 \cdot 10^{-4}$ to $10^{-2}$ m. These droplets are falling and therefore the fuel film is formatted as is shown in Fig. 4. This film slowly spreads across the floor of the building and flows out of the hole after impact.

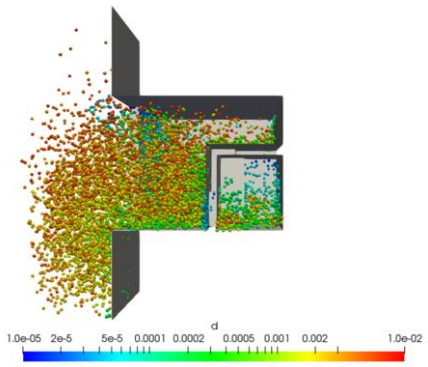


**Fig. 2.** Position of spray particle for nonreacting flow in time t = 20 s.

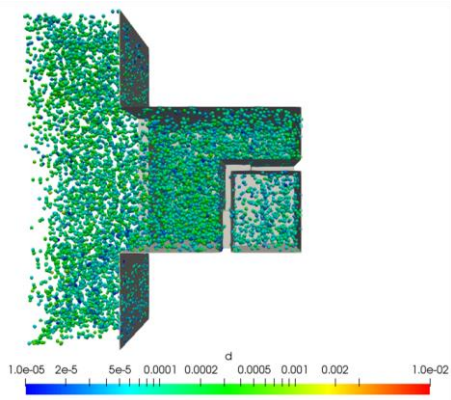


**Fig. 3.** Position of spray particle for reacting flow in time t = 20 s.

The formation of a fuel film is not observed in the case of the reacting flow field. Instead the "fireball" is predicted by simulation as is possible to see in Fig. 5 where is shown isosurface of temperature at value 800 K. This "fireball" is caused by the burning of all droplets in computational domain. This explains lower diameter of droplets which are observed in Fig. 3.

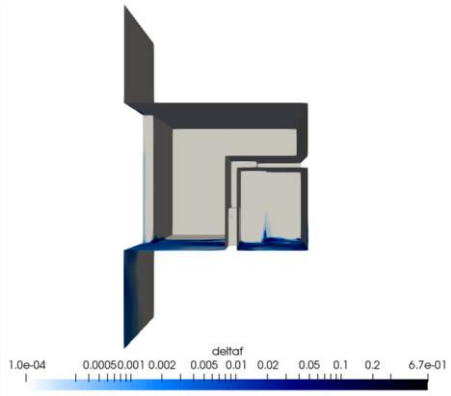


**Fig. 4.** Formation of fuel film in the case of nonreacting simulation in time t = 60 s.

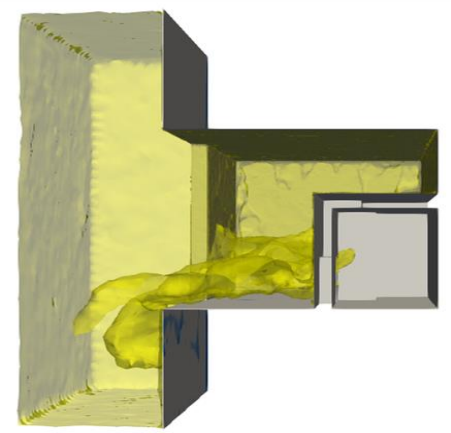

**Fig. 5.** Isosurface of temperature (800 K) in reacting case in time t = 60 s.

If similar boundary conditions are used on different computational geometry with the large open area, then different results are observed. Changes are visible on droplets diameter where the value between and 5e-3 m and 5e-2 m is observed (Fig. 6.).

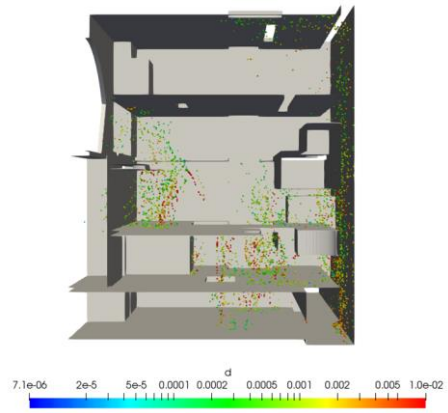


**Fig. 6.** Position of spray particles for flow time t = 20 s.

Also, the formation of fuel film is observed which flows into the basement (Fig. 7.) where the fuel is accumulated and slowly burns out (Fig. 8.).

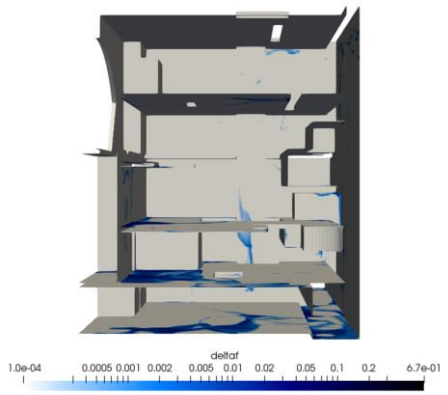


**Fig. 7.** Formation and position of fuel film in large open area in time t = 60 s.

Thus, due to fire can affect the storage area where flammable liquids can be placed in a building and intensify the fire. At the same time, considerable amounts of smoke will be generated and spread through the building area.

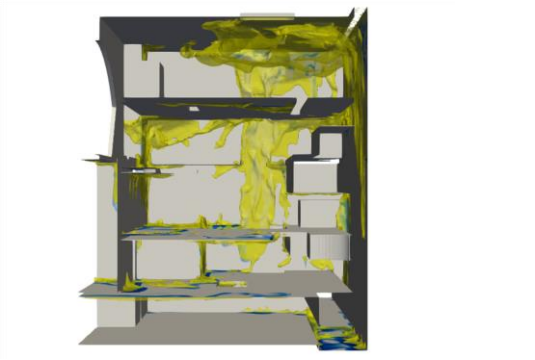

**Fig. 8.** Isosurface of temparature (800 K)in large open area in time t = 60 s.

## 4 Conclusion

Results of a fire simulation after an aircraft crashes into a building are shown in this paper. These results are burdened with the assumptions that are applied. These assumptions relate to neglecting the influence of debris, infiltration of fuel and breaking the airplane. If these simplifying assumptions were not applied then the complexity of simulation and hardware and software resources increase significantly.

Two different scenarios of fire propagation are indicated in the results. If the fuel is sprayed into close space, then it is combusted in the form of aerosol and fuel film is not created. Compared to the case, where the fuel is sprayed into open space then the only ¼ of fuel is burned in the form of aerosol and fuel film is formed from the rest of the fuel.

Simulations are mostly limited by using the simplest combustion model. If a more complex combustion model is used, then it is expected improvement results. Especially a soot and CO production is captured. For this purpose, it is a necessary modified combustion model which was used in previous work [1] and it was optimized for combustion of liquid fuels in jet engines.

## Acknowledgments

This work has been supported by the grant project of the Ministry of the Interior of the Czech Republic no. VI20172020092 and by The Ministry of Education, Youth and Sports from the Large Infrastructures for Research, Experimental Development and Innovations project „IT4Innovations National Supercomputing Center – LM2015070“.

## References

1. V. Betak, J. Kubata, *Proceedings of the International Conference Experimental Fluid Mechanics* (2015)
2. F. R. Menter, F. Kuntz, R. Langhtry, Turb., He. Ma. Trans. (2003)