Numerical simulation of the gas flow with radiation and heat transfer

Jan Česenek[1,*], *Martin* Kyncl[1], and *Jaroslav* Pelant[1]

[1]VZLU - Czech Aerospace Research Center, Beranových 130, 199 05 Praha - Letňany, Czech Republic

Abstract. In this article we focus on the heating of the solid body placed in the vicinity of the intense heat source. We simulate the heat transfer caused by the direct contact with surrounding heated gas. Furthermore it is necessary to take into account the heat transfer caused by the radiation, which radically affects the resulting heat flux. We work with the non-stationary viscous compressible fluid flow with additional heat sources, described by the RANS equations. The heat radiation is simulated using other supplemented equations. The heat transfer equations are solved inside the considered body.

1 Introduction

CFD methods are used in many industrial applications. In some application it is not only important to study influence of the flow around obstacle but it is also necessary to study the heat transfer. In this article we will deal with impact of the heat source to the surroundings in particular to the solid wall. We will simulate heat transfer by the conduction, convection and also by the radiation transfer. For this type of problems we have two regions. First region is occupied by the fluid. This part is described by the compressible Navier-Stokes equation. Second region is considered as a solid wall. In this part heat equation is used for the description of the heat flux through such wall. Between this two regions we have to use suitable boundary conditions for heat transfer. We will present some simulations with two types of heat source. In the first case it will be hot cube and in the second the heat will be released from chemical reaction. The results from the simulations will be shown.

2 Formulation of the $k-\omega$ turbulence model

Compressible flow is described by the system of Navier-Stokes equations:

$$\frac{\partial \mathbf{w}}{\partial t} + \sum_{s=1}^{3} \frac{\partial \mathbf{f}_s(\mathbf{w})}{\partial x_s} = \sum_{s=1}^{3} \frac{\partial \mathbf{R}_s(\mathbf{w}, \nabla \mathbf{w})}{\partial x_s}, \tag{1}$$

where for $s = 1, ...,3$ we have

$$\begin{aligned}
\mathbf{w} &= (w_1, ..., w_5)^T = (\rho, \rho v_1, ..., \rho v_3, E)^T, \\
\mathbf{f}_s(\mathbf{w}) &= (f_{s,1}, ..., f_{s,5})^T \\
&= (\rho v_s, \rho v_1 v_s + \delta_{1s} p, ..., \\
&\quad \rho v_3 v_s + \delta_{3s} p, (E+p) v_s)^T, \\
\mathbf{R}_s(\mathbf{w}, \nabla \mathbf{w}) &= (R_{s,1}, ..., R_{s,5})^T \\
&= \Big(0, \tau^V_{s1}, ..., \tau^V_{s3}, \\
&\quad \sum_{r=1}^{3} \tau^V_{sr} v_r + \left(\frac{c_p \mu_L}{Pr}\right) \frac{\partial \theta}{\partial x_s}\Big)^T,
\end{aligned}$$

where

$$\begin{aligned}
\tau^V_{sr} &= -\frac{2}{3} \mu_L \operatorname{div} \boldsymbol{v}\, \delta_{sr} + 2\mu_L\, d_{sr}(\boldsymbol{v}), \\
d_{sr}(\boldsymbol{v}) &= \frac{1}{2}\left(\frac{\partial v_s}{\partial x_r} + \frac{\partial v_r}{\partial x_s}\right).
\end{aligned}$$

We use the following notation: $\boldsymbol{v} = (v_1, ...,v_3)$ - velocity, ρ - density, p - pressure, θ - absolute temperature, E - total energy, γ - Poisson adiabatic constant, κ - heat conduction coefficient, c_v - specific heat at constant volume, c_p - specific heat at constant pressure, where $c_p = \gamma c_v$, Pr is the laminar Prandtl number, which can be express in the form $Pr = \frac{c_p \mu_L}{\kappa}$, μ_L is the dynamic viscosity coefficient dependent on temperature via Sutherland's formula. The above system is completed by the thermodynamical relations

$$\begin{aligned}
p &= (\gamma - 1)\left(E - \frac{1}{2}\rho\, |\boldsymbol{v}|^2\right), \\
\theta &= \frac{1}{c_v}\left(\frac{E}{\rho} - \frac{1}{2}\, |\boldsymbol{v}|^2\right)
\end{aligned}$$

*e-mail: cesenek@vzlu.cz

To this system of equations we have to add equation describing radiation model

$$\nabla \cdot (\Gamma \nabla G) - aG \quad = \quad -4\epsilon\sigma T^4 - E$$

where G is the so-called irradiation, which can be expressed as $G := \nabla q_r$, where q_r is heat flux $[qr] = Wm^{-2}$, absorbivity a, emissivity ϵ and E, which charaterize volume radiation source. In the solid region we solve the heat equation.

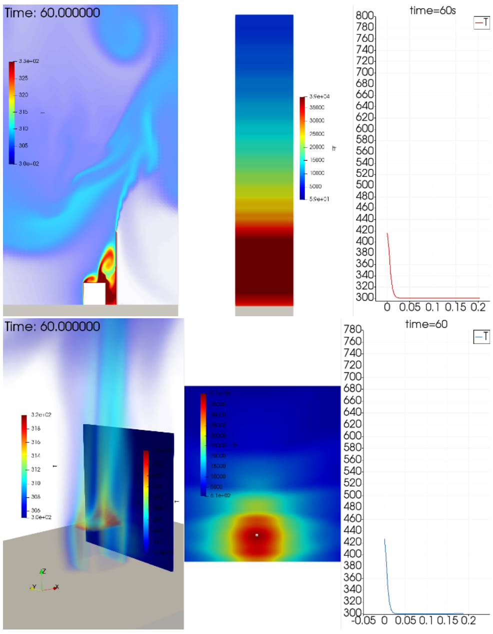

Fig. 1. Heated cube, simulation in 2D and 3D. (From left) The air temperature, the heat flux between the wall and the cube and the distribution of the temperature through the heated wall at time instance 60s.

3 Numerical experiments

In this section we present computation results. For the simulations we have used *chtMultiRegionFoam* solver from the open source code OpenFoam.

In the first example the source of the heat is presented by very hot solid cube, which has temperature 1200K. Surrouding air and wall has temperature 300K at the beginnig of the simulation. This hot cube radiates heat to the surroundings and warms up the solid wall, which is at distance 1.5m from the cube. These simulations were carried out on fine mesh in 2D, and on the coarser mesh in 3D. Figures (1) - (2) show the results of the temperature and the heat flux at two different time instances. The total heat flux between hot cube and wall surface is slowly diminishing as the wall heats up.

In further simulations we used the combustion (and radiation) as the source of the heat. Due to the large complexity of the proper combustion simulation we reduced chemical reactions to the following equation:

$$\begin{aligned} & C_{12}H_{23} + 28.75O_2 + 107.8125N_2 \\ \rightarrow \quad & 23CO_2 + 11.5H_2O + 107.8125N_2, \end{aligned}$$

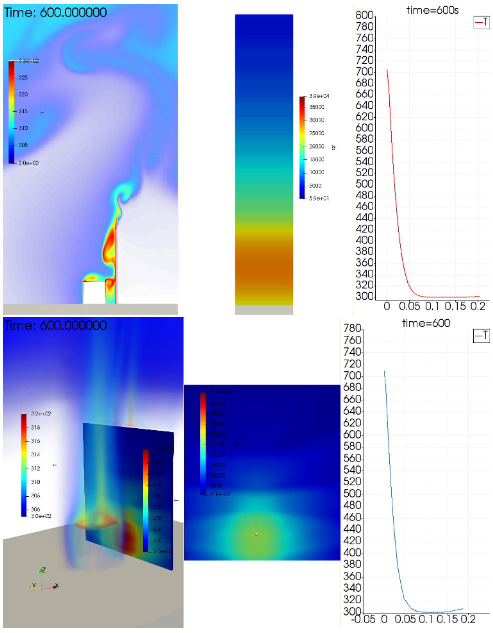

Fig. 2. Heated cube, simulation in 2D and 3D. (From left) The air temperature, the heat flux between the wall and the cube and the distribution of the temperature through the heated wall at time instance 600s.

where $C_{12}H_{23}$ is the fuel. Figure (3) shows the 2D and 3D simulation of the combustion with prescribed fuel mass flow 10 $g \cdot m^{-2} \cdot s^{-1}$ at the selected area $3.0m \times 3.0m$. Figure (4) shows the simulation with concrete wall (d=0.2m, h=10m) placed 1.5m far from this source. The fire gets attached to the wall in 2D simulation, and the wall is the heated up.

For further simulations we set the concentration of the fuel to 0.5 in the area $2.0m \times 2.0m \times 0.1m$, which is placed 1.5m far from the wall. This presents the pool of the fuel, where the concentration is fixed during the whole simulation. We studied the chemical reaction and its heat and radiation impact to the surround area which is occupied by the air and heat transfer to the solid floor and concrete wall. Figures (5) - (6) show the results of the temperature and heat flux distribution at two different time instances.

Last example is the 3D simulation of the previous case with the fuel in volume 2.0m x 2.0m x 0.5m situated 1.5m far from the wall. The concentration fraction of the fuel was set to 0.5. Here we used coarser meshes than in the 2D cases. Presented figures (7) resp. (8) show the distribution of the temperature on the solid wall and floor resp. in the air, figure (9) shows the concentration of the fuel and CO_2. Such approach can be used for the simulation of the poured burned fuel in some real area.

4 Conclusion

In this paper we dealt with the radiation and its impact to the surrounding area. Presented results show, that it is possible to simulate poured burning fuel by a pool filled with the fuel. Here the large amount of heat is transferred by the radiation, and therefore it is necessary to include some radiation model for proper practical application.

Acknowledgements. The results originated with the support of Ministry of the Interior of the Czech Republic, project VI20172020092 - Simulation of fire and smoke dispersion in critical infrastructure area in case of accident or aircraft attack

References

1. M.B. Hermosilla, *Study of heat and mass transfer applications in the field of engineering by using openfoam*, Bachelor's Thesis, (2016)
2. J. Česenek, *EPJ WoC* **67**, 02015 (2014)
3. J. Česenek, *EPJ WoC* **180**, 02016 (2018)
4. J. Česenek, *EPJ WoC* **114**, 02012 (2016)
5. J. Česenek, *EPJ WoC* **143**, 02014 (2017)
6. J. Česenek, *EPJ WoC* **213**, 02011 (2019)
7. M. Kyncl and J. Pelant, *EPJ WoC*, **45**, 01054, (2013)
8. M. Kyncl and J. Pelant, *EPJ WoC*, **180**, 02052, (2018)
9. M. Kyncl and J. Pelant, *EPJ WoC* **213**, 02050 (2019)
10. M. Kyncl and J. Pelant, *EPJ WoC* **213**, 02051 (2019)
11. M. Kyncl and J. Pelant, *EPJ WoC* (to be published) (2020)

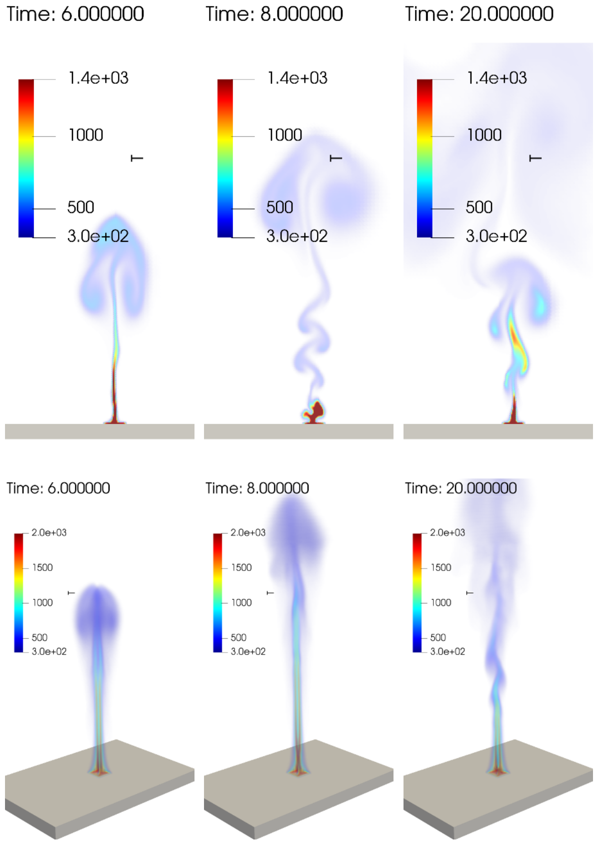

Fig. 3. Simple combustion simulation in 2D (31536 elements) and 3D (834616 elements), temperature field at various time instants. Constant fuel source fixed to 10 $g \cdot m^{-2} \cdot s^{-1}$.

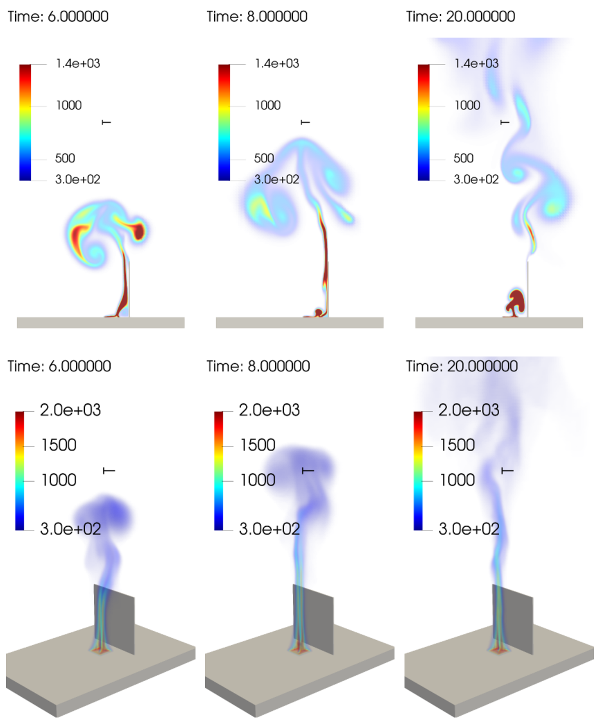

Fig. 4. Combustion simulation in 2D (124223 elements) and 3D (834616 elements), temperature field at various time instants. Constant fuel source fixed to 10 $g \cdot m^{-2} \cdot s^{-1}$. In 2D case the fire becomes attached to wall.

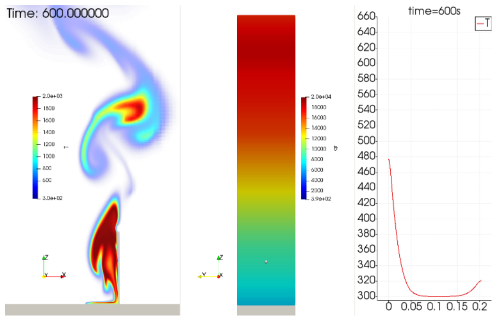

Fig. 5. 2D combustion simulation. (From left) The air temperature, the heat flux through the wall surface, and the distribution of the temperature through the heated wall at h=1.5m, time instance 600s.

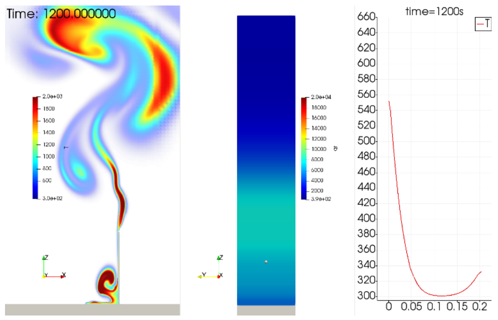

Fig. 6. 2D combustion simulation. (From left) The air temperature, the heat flux through the wall surface, and the distribution of the temperature through the heated wall at h=1.5m, time instance 1200s.

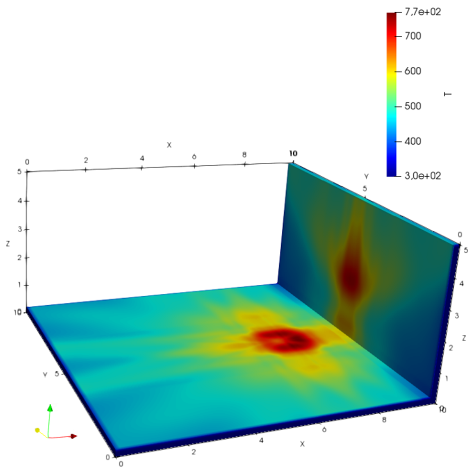

Fig. 7. The distribution of the temperature on the wall and floor.

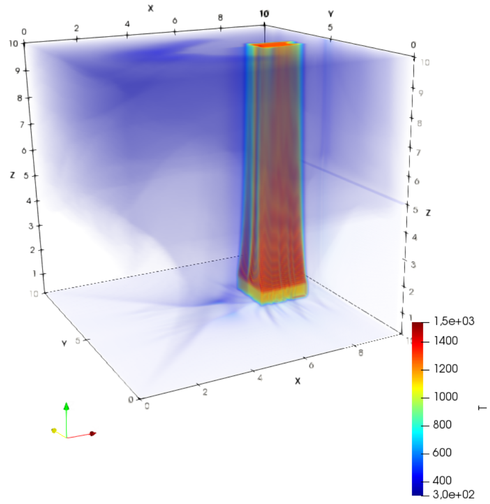

Fig. 8. The distribution of the temperature in the air.

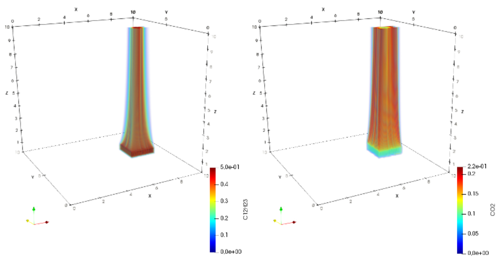

Fig. 9. The distribution of the concentration of the fuel (left) and CO_2 (right).

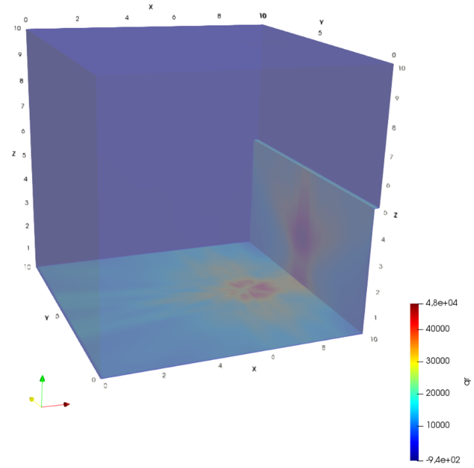

Fig. 10. The distribution of the heat flux.