26th International Conference
ENGINEERING MECHANICS 2020
Brno, Czech Republic, November 24 – 25, 2020

NUMERICAL SIMULATIONS OF THE HEAT TRANSFER BETWEEN GAS AND SOLID

Česenek J.[*], Kyncl M.[**], Pelant J.[***]

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 URANS equations. The heat radiation is simulated using other supplemented equations. The heat transfer equations are solved inside the considered body.*

Keywords: Compressible gas flow, Combustion, URANS, Heat transfer, Radiation, Finite volume method, Software, 3D.

1. Introduction

The main topic of this work is to estimate the raise of the temperature in the vicinity of the concrete construction caused by the fire source. This complex task requires the simulation of the combustion, compressible gas flow, heat conduction, and radiation heat transfer. The physical theory of the compressible fluid motion is based on the principles of conservation laws of mass, momentum, and energy. The mathematical equations describing these fundamental conservation laws form a system of partial differential equations, the so-called Navier-Stokes equations with k-omega turbulence model. Here we assume the flow of reacting gas mixture in the gravitational field, which gives us additional differential equations simulating combustion and mass conservation of additional gas species.

$$\frac{\partial w}{\partial t}+\sum_{s=1}^{3}\frac{\partial f_s(w)}{\partial x_s}=\sum_{s=1}^{3}\frac{\partial R_s(w,\nabla w)}{\partial x_s}$$

Here $w(x,t)$ is the state vector, f_s are the inviscid fluxes, R_s are the viscous fluxes, S is the source-term vector.

Suitable boundary conditions are used to describe the heat transfer between the gas, solid region, and heat source (thermal radiation). The heat transfer equation is solved inside the solid region.

Due to the large complexity of the proper combustion simulation we reduced chemical reactions to necessary minimum. A simple chemistry for the simulation of combustion can be written as

$$C_{12}H_{23}+28.75O_2+107.8125N_2=23CO_2+11.5H_{2O}+107.8125N_2,$$

where $C_{12}H_{23}$ is the fuel.

The finite volume method with implicit time marching is used for the discretization of the partial differential equations on unstructured meshes. In this research the open source code OpenFOAM was used to simulate various scenarios.

[*] RNDr. Jan Česenek, PhD.: Výzkumný a zkušební letecký ústav, a. s, Beranových 130; 199 05, Prague; CZ, cesenek@vzlu.cz
[**] RNDr. Martin Kyncl, PhD.: Výzkumný a zkušební letecký ústav, a. s, Beranových 130; 199 05, Prague; CZ, kyncl@vzlu.cz
[***] RNDr. Jaroslav Pelant, CSc.: Výzkumný a zkušební letecký ústav, a. s, Beranových 130; 199 05, Prague; CZ, pelant@vzlu.cz

doi: 10.21495/5896-3-098

2. Numerical experiments

Here we assume the source of the heat formed by the hot cube with dimensions 3 x 3 x 3 m and temperature 1200 K. The heat transfer to adjacent solid wall is caused by the thermal radiation and by the temperature of the adjacent gas. The initial temperature in the whole computational area was set to 300 K. The simulated solid concrete wall was 10m high, at the distance of 1.5 m from heater. The computational mesh in 2D consisted of 103 775 elements, the 3D mesh of 567 034 elements. Figs. 1 and 2 show the temperature distribution, radiative flux source at the wall, and temperature distribution inside the wall (cut at h = 1.5 m) at chosen time instants 60 s and 600 s.

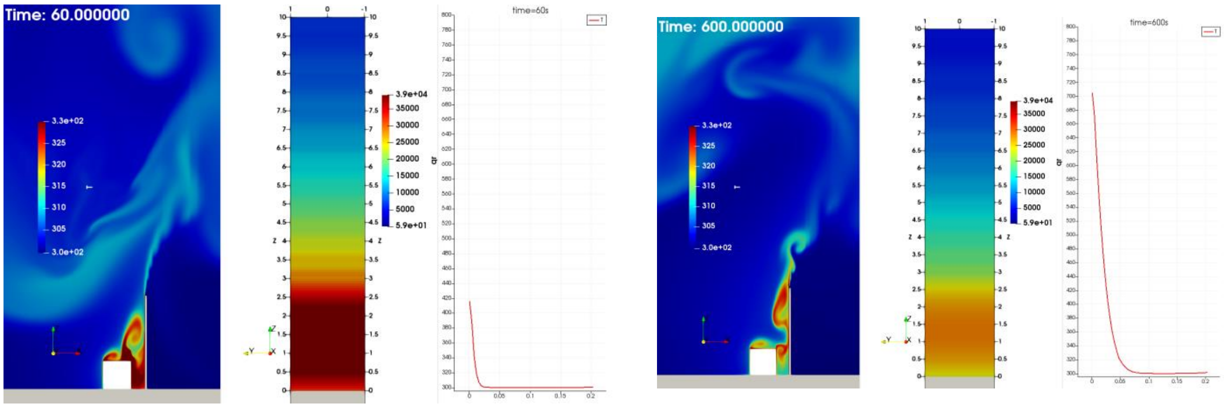

Fig. 1: The 2D simulation of the heat transfer. The source of the heat is a hot cube with temperature 1200 K. The temperature of the solid wall raises with time.

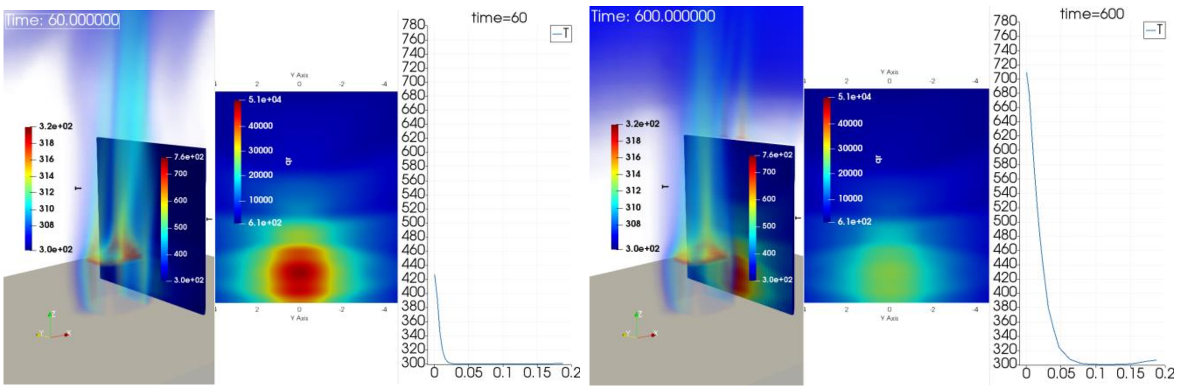

Fig. 2: The 3D simulation of the heat transfer. The source of the heat is a hot cube with temperature 1200 K. The temperature of the solid wall raises with time.

Further we simulated the combustion as the source of the heat. The fuel mass flow was prescribed to 10 $g \cdot m^{-2} \cdot s^{-1}$ at the selected area 3.0 m x 3.0 m. Fig. 3 shows the computational results of the non-stationary simulation.

In another simulation a pool of the fuel was set in front of the solid wall, where the fuel concentration is fixed to 0.5 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. Fig. 4 shows the results of the temperature field at given time instant.

Such approaches can be used for the simulation of the poured burned fuel in some real area.

3. Conclusions

In this article we worked with the heat transfer between gas and the solid. We assumed the heat sources caused by the heated cube, and by the combustion of the fuel. The large amount of heat is transferred by the radiation, and therefore it is necessary to include some radiation model for the practical application. The numerical examples were presented.

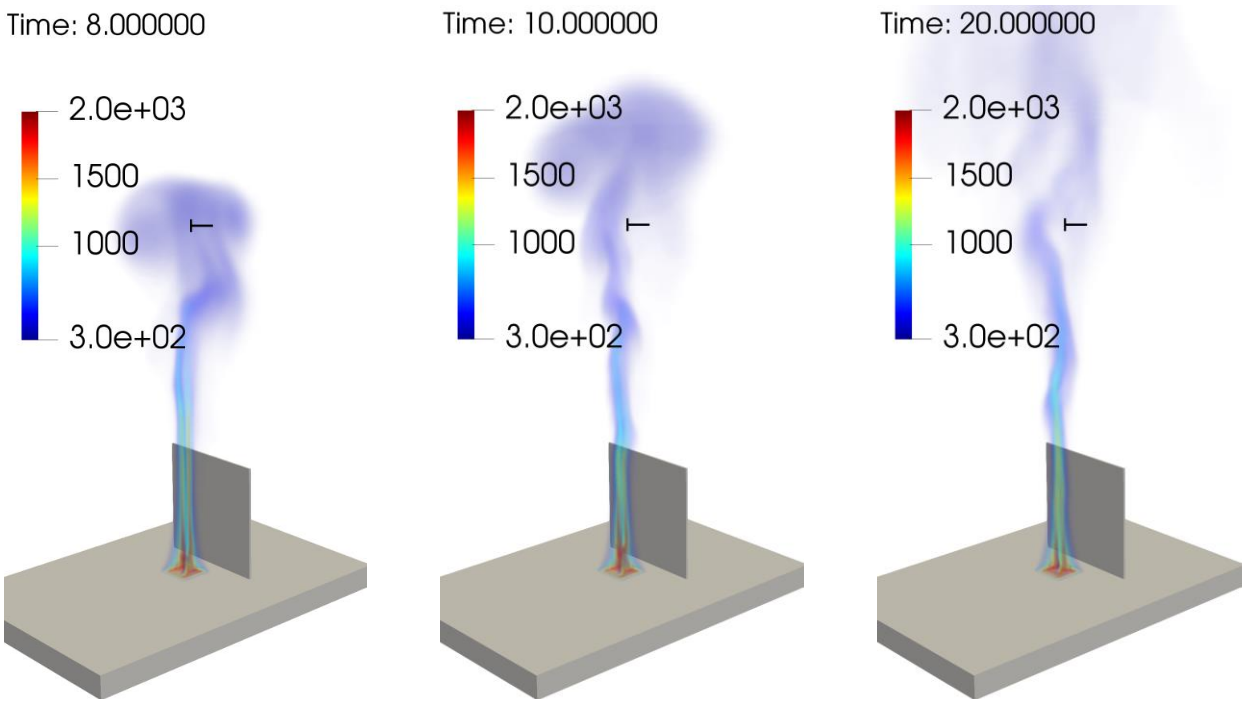

Fig. 3: The 3D simulation of the combustion. Temperature field at various time instants. Constant fuel source fixed to 10 $g \cdot m^{-2} \cdot s^{-1}$.

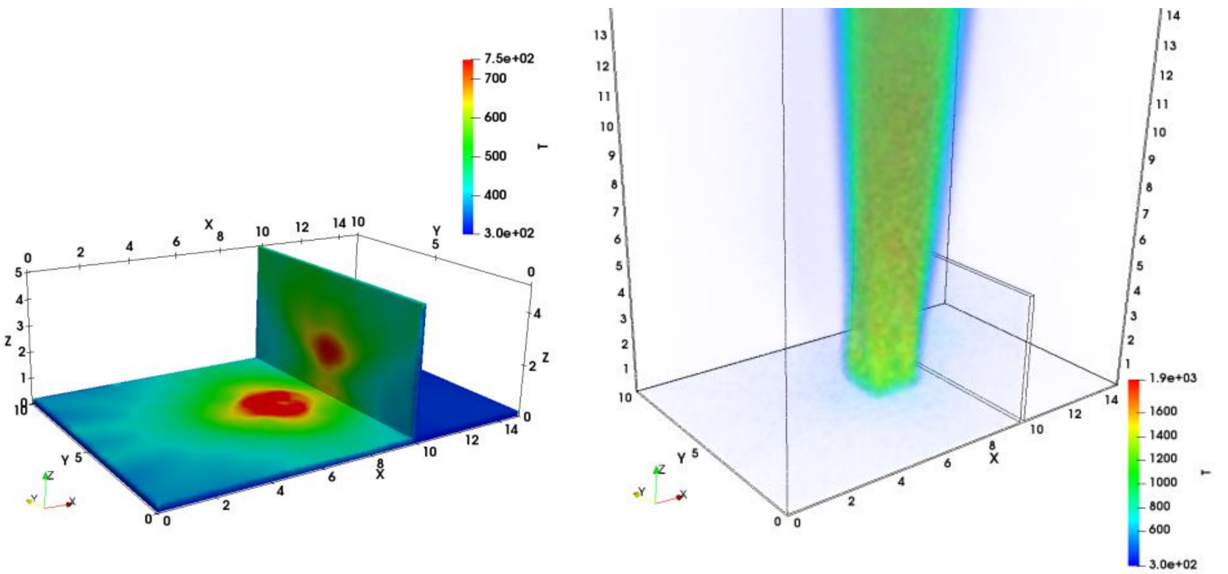

Fig. 4: The 3D simulation of the heat transfer. The distribution of the temperature on the wall(left) and the distribution of the temperature in the gas.

Acknowledgement

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

Wilcox, D. C. (1998) Turbulence Modeling for CFD, California, USA.

Kok, C. J. (2000) Resolving the dependence on free-stream values for k-omega turbulence model, in: AIAA Journal, vol. 38., no. 7.

Kyncl, M. and Pelant, J. (2012) Implicit method for the 3D RANS equations with the k-w (Kok) Turbulent Model, Technical report R-5453, VZLU, Beranovych 130, Prague.

Feistauer, M. (1993) Mathematical Methods in Fluid Dynamics, Harlow, England.

Toro, E.F. (1997) Riemann Solvers and Numerical Methods for Fluid Dynamics, Berlin, Germany.

Feistauer, M., Felcman, J., and Straškraba, I. (2003) Mathematical and Computational Methods for Compressible Flow, Oxford, England.

Kyncl, M. (2011) Numerical solution of the three-dimensional compressible flow, Doctoral Thesis, Prague.

Česenek, J., Kyncl, M. and Pelant, J. (2019) Numerical simulation of the gas flow with radiation and heat transfer. EFM 2019.

Kyncl, M., Česenek, J. and Pelant J. (2019) Numerical simulation of the mixture flow over terrain wave. EFM 2019.