COMPUTATIONAL MECHANICS 2019
35th conference with international participation
Srní
November 4 - 6, 2019

Comparison of two numerical approaches for simulation of fire with radiative heat transfer into solid walls

V. Běták[a]

[a] Czech Aerospace Research Centre, Department of Engines, Beranovych 130, 199 05 Prague, Czech Republic

1. Introduction

Mathematical modeling of fluid flow is important in modern engineering. It allows us to improve performance characteristics and increase the lifetime of modern products due to the prediction of drag, lift, pressure and thermal load, pollutant emission or study of highly unsteady transition phenomena. Mathematical modeling is often used in cases where the application of the experimental method is limited such as in the case of combustion. Based on the length scale, these simulations can be divided into three areas. The smallest one relates to the simulation of combustion in the combustion chamber of gas turbines [1]. The middle one relates to modeling combustion in industrial chambers such as glass melting furnace [2]. The great one is focused on fire modeling in buildings and exteriors. Part of this topic will be discussed in the following paper.

There are two possibilities for fire modeling. The first one is based on an empirical approach and does not include any additional equation connected with combustion modeling. The second approach is based on combustion modeling and it is suitable for complex geometries.

To evaluate the fire resistance, the heat load of the wall must be overstated. The wall can be heated by direct contact with the hot gases or by radiation. The radiation model and its dependence on the combustion model will be presented in this paper.

2. Mathematical model

The fire modeling is based on the following system of equations

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_j) = 0, \quad (1)$$

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

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

where ρ is density, u_i is a component of the velocity vector, p is pressure, μ is a dynamic viscosity, h is enthalpy, α is a coefficient of heat diffusion and μ_t is a turbulent viscosity. Source terms in enthalpy equations represent via S_h.

The empirical model is based on the combustion heat $\dot{Q}$ of fuel and relationship defined in [3,5] especially length of flames which is based on the diameter of the fire pool

$$L_f = -1.02D + 0{,}0148\dot{Q}^{\frac{2}{5}}. \quad (4)$$

These parameters define a cell zone where is prescribed source term in enthalpy equations.

If the combustion model is included, then the system of equations (1) – (3) is extended by transport equation for mass fraction Y_i

$$\frac{\partial}{\partial t}(\rho Y_i) + \frac{\partial}{\partial x_j}(\rho Y_i) = \frac{\partial}{\partial x_j}[(\mu + \mu_t)\frac{\partial Y_i}{\partial x_j}] + S_{Y_i}, \qquad (5)$$

where S_{Y_i}represent source terms.

The Number of additional PDE and ODE equations is depended on the complexity of the combustion model. In this case is used the simplest one equation combustion model called "infinitelyFastChemistry" [6] which is used one constant $C$for tuning heat release rate.

The radiative model has to be included in fire simulation. The P1 radiation model [7] was chosen first but had to be replaced by DO(discrete-ordinates) radiative model [4].

In order to capture the heat flux into the wall correctly, the energy equation in solid region has to be solved. This equation is described by following partial differential equation

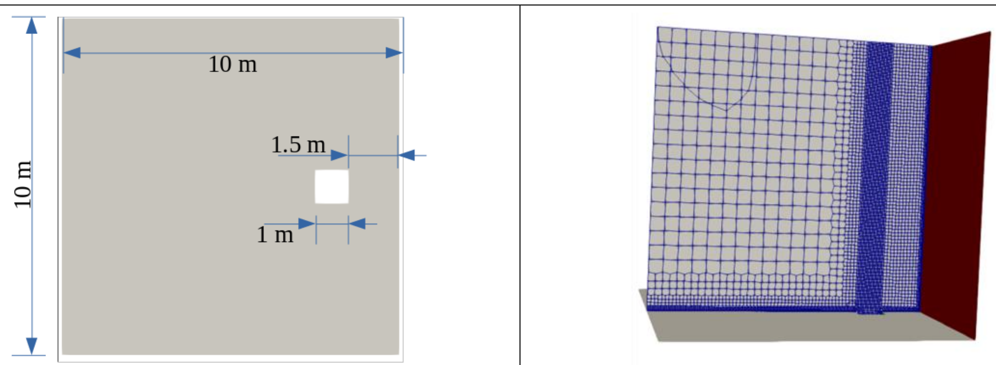
$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x_j}[(\alpha)\frac{\partial h}{\partial x_j}] = 0\,. \qquad (6)$$

Based on required properties a suitable solver has to been chosen. ChtMultiRegion from OpenFOAM [6] library is used. This solver is based on PISO algorithm and allow to solve coupled problem of fluid flow with heat transfer through solid region.

3. Results

Fire simulation is tested on simplified geometry with a defined pool (1x1 m) which is 1.5 m away from the wall as is shown in Fig. 1. The rate of fuel (kerosene) vaporization from the pool is prescribed as 0.022 kg s^{-1} m^{-2} at temperature 480 K. This amount of fuel corresponding to firepower 0,96 MW. The ambient boundary conditions (300 K, 101325 Pa) is prescribed at free stream boundaries.

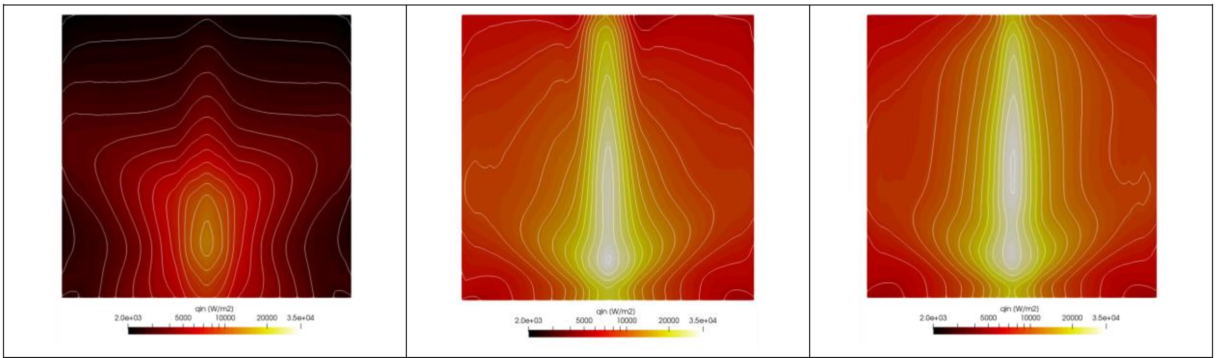
The simulation model is shown in Fig. 1 where the pool is located at a given distance from the wall. The computational domain is constructed by hexa-dominant algorithm and consist from 488k cells.



a) geometry description | b) example of computational domain

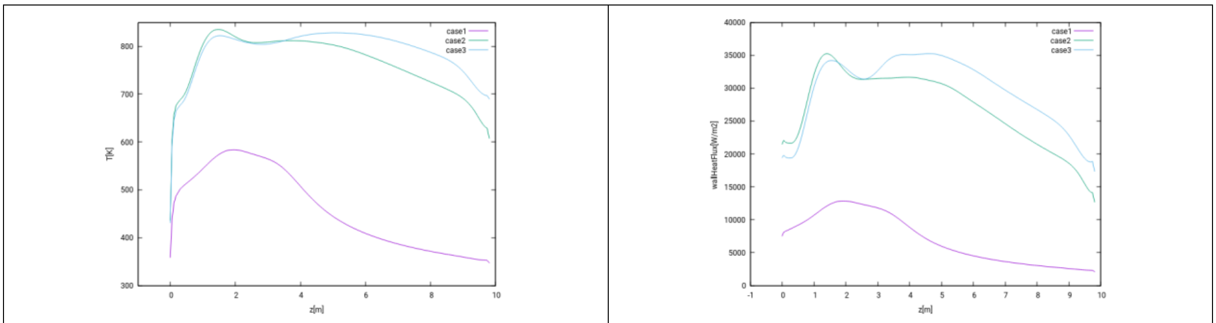
Fig. 1. The computational domain

Determining radiative heat flux is the goal of the simulation. There are compared three models. The first one is based on empirical approach and RANS turbulence model (*"case1"*). The second and third one is based on a simplified combustion model which is used with different turbulence model. The first one is based on RANS approach (*"case2"*) and the second one is based on DES approach (*"case3"*). The results are shown in Figs. 2 and 3. It is possible to see that the model based on a simplified chemical model predicting about three times greater radiative heat flux. This is caused by the application of the combustion model and modeling of radiative heat flux from three atomic molecules of H_2O and CO_2. The difference between the turbulent model is at a height above 3 m where the higher radiative heat flux and wall temperature are predicted by simulation based on DES turbulence model.



a) case1	b) case2	c) case3

Fig. 2. Comparison of prediction radiative heat flux into wall by diferent model



a) temperature	b) wall heat flux

Fig. 3. Profile of selected variables along wall symmetry axis

The radiative heat flux intensity can be influenced by setting the combustion model constant C. If the value is reduced, then radiative heat flux and wall temperature increased. If, on the other hand, the value is increased then radiative heat flux decreased. To achieve a similar flame temperature as in the case of the empirical model, it is necessary to select a constant in the range from 20 to 50 as is possible to see in Table 1.

Table 1. Overview of simulations

	Combustion	**Turbulence**	**C**	**Wall heat flux [MW]**	**Wall average temperature [K]**	**Wall maximum temperature [K]**	**Flame maximum temperature [K]**
Case 1	Emp.	RANS		0.394	392	583	1405
Case 2	Inf.Fast	RANS	5	1.088	533	835	1597
Case 3	Inf.Fast	DES	5	1.183	552	829	1631
Case 4	Inf.Fast	DES	2	1.186	558	840	1711
Case 5	Inf.Fast	DES	1	1.215	556	840	1739
Case 6	Inf.Fast	DES	0.5	1.251	556	843	2269
Case 7	Inf.Fast	DES	0.1	2.283	709	1059	>2500
Case 8	Inf.Fast	DES	10	1.201	547	823	1584
Case 9	Inf.Fast	DES	20	1.174	546	814	1589
Case 10	Inf.Fast	DES	50	0.855	498	709	1320
Case 11	Inf.Fast	DES	100	0.744	460	640	1208

4. Conclusion

Validation and calibration of a simplified combustion model is shown in this paper. This model is designed to simulate fire and temperature flux into a building wall for a study fire and structural safety of buildings. It is necessary to choose a combustion model that works with three and more atomic gases. These gases have a significant influence on the determination of the radiation heat flux. Their neglect reduces the radiation flow to 1/3.

The calibrated model is used to simulate large fires near and inside buildings.

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] Betak, V., Numerical simulation of combustion in small aero engines by using segregated algorithm,Proceedings of the 12th conference on Power System Engineering, Thermodynamics & Fluid Flow, Pilsen, 2019.

[2] Betak, V., Advanced DES simulations of oxy-gas burner located into model of real melting chamber, Proceedings of the 14th international seminar on furnace design, operations and processes simulation, Velke Karlovice, 2017.

[3] ČSN EN 1991-1-2.

[4] Fiveland, W. A., Three-dimensional radiative heat-transfer solutions by the discrete-ordinates method. Journal of Thermophysics and Heat Transfer 2 (4) (1988) 309-316.

[5] Kucera, P., Pokorny, J., Fire dynamics, VŠB-TUO, 2014. (in Czech)

[6] OpenFOAM, www.openfoam.com

[7] Sazhin, S.S., Sazhina, E.M., Faltsi-Saravelou, O., Wild. P., The P-1model for thermal radiation transfer: advantages and limitations, Fuel 75 (3) (1996) 289-294.