Computational prediction of fire spread to a solid material with ANSYS CFX

Andrej Horvat
Intelligent Fluid Solutions Ltd.
127 Crookston Road
SE9 1YF, London, United Kingdom
Tel./Fax : +44 (0)1235 819 729
E- mail: andrej.horvat@intelligentfluidsolutions.co.uk

Yehuda Sinai
ANSYS Europe Ltd.
West Central 127
Milton Park
OX14 4SA, Abingdon
United Kingdom

Abstract

A modelling approach was developed to simulate fire spread from combusting gases to a solid surface material. The development work was performed at ANSYS Europe Ltd. as a part of the EC funded Firenet project, which investigated phenomena related to under-ventilated fires.

The motivation to develop a fire spread model and to link it to the ANSYS CFD package was the need to obtain a reliable computational tool for solid material ignition under fire conditions, subsequent pyrolysis and combustion. The objective was to develop a fire spread model that could be linked to the ANSYS CFD package and hence used in fire safety engineering as well as in industrial applications (e.g. for combustion of solid fuels). Alternative approaches to this modelling are also being explored at ANSYS Europe Ltd.

The computational model was validated by using experimental data from CNRS-ENSMA-Poitiers. In their case, a propane burner with thermal power of 55 kW was used as a primary source of fire and square beech wood samples (30 mm × 30 mm × 5 mm) as fire spread targets. The comparison of the calculated and the experimentally measured temperatures shows a good agreement for solid material samples closer to the main fire source where the model predicts the ignition time very well. For the samples further away from the main fire source, the differences are larger (the authors believe due to geometrical simplifications), which are discussed in the paper. In general, the performance of the developed fire spread model is satisfactory although caution is needed in its application due to sensitivity to material properties and geometrical simplifications.

1 Introduction

A fire in a confined space can be divided into three distinct stages based on fuel and oxygen consumption, heat release and variation of average gas temperature. These stages are usually described as the fire growth period, fully developed fire and decay period. Flashover is a short period of transition from a localized initial fire to the fully developed fire where all fuel surfaces within the compartment start to burn [1]. During the pre-flashover stage, the fire develops from its origin, forming a hot layer of combustion products below the ceiling of the enclosure. Thermal radiation from the fire and the hot layer raises the surface temperature of

the surrounding combustible material. The material starts to disintegrate chemically as well as thermally [2] releasing volatiles that may ignite. The ignition of the combustible volatiles results in a rapid flame spread from a localized fire to all combustible surfaces. A more detailed description of this phenomenon may be found in [1], [3] and [4].

From the fire-fighting perspective, flashover is a critical stage of fire growth. Specifically, when flashover takes place, the probability of survival of occupants decreases rapidly. As the transition from the initial localized fire to the general conflagration takes usually less than a minute [5], fatalities are very likely to occur. Also, flashover creates a large increase in the rate of combustion; therefore, significantly greater effort is needed to reduce the burning material below its ignition temperature [6].

Due to the hazard associated with flashover, the subject has received a fair amount of attention in the literature. Drysdale [1] presents arguably the most comprehensive overview of experimental studies. More recent analysis of flashover experimental data can be found in [5] and [7]. Whilst the emphasis of this paper is on computational modelling of flashover, it is also important to mention full-scale experiments conducted by White et al. [8], in which a train fire was allowed to become fully developed, involving all combustible materials within the train.

The present paper describes modelling work performed by ANSYS Europe Ltd. to develop a reliable computational tool for solid material ignition under fire conditions, subsequent pyrolysis and combustion. The work was conducted as a part of FireNet project, which was sponsored by EU Commission. Comparing the collected experimental and simulation results, the paper discusses suitability of the considered modelling technique, analyses weaknesses of the model and gives recommendations for further development. It also briefly outlines variety possible application areas for the developed fire spread model.

2 Modelling approach

Computational models used to analyse flashover can be classified into zone and field models. The theoretical background of zone models is the conservation of mass and energy in fire. The simulation domain is divided into a small number of separate zones and the conditions in each zone are assumed to be constant. Most often the fire is described with two-zones (lower cold layer and upper hot layer). As such description of a complex phenomenon is rather coarse, zone models have to incorporate empirical observations regarding fire dynamics and smoke movement. Some of the applications of zone models to flashover scenario were published by Spearpoint et al. [4], Lou et al. [9], and Chow [10]. More recently, Novozhilov [11] presented an analysis of flashover development under fire suppression conditions using a zone model.

In comparison to zone models, field models offer much greater modelling flexibility due to local, Eulerian field description of physical variables. Initial applications of field models to fire spread over solid surfaces were published by Atreya [12] and Fredland [13]. Nevertheless, a numerical prediction of transient behaviour of flashover that incorporates flow dynamics, convective and radiation heat transfer, thermal and chemical decomposition of solid material remains a challenging task. Although, more recent applications [9, 14, 15] show better prediction of magnitude and trends during flashover, large discrepancies in comparison to experimentally obtained values still exist [16].

In the present work, a numerical field model of fire spread from a primary fire onto flammable solid targets has been developed. The geometry of the simulation domain and the model parameters follow closely the experiments conducted at CNRS-ENSMA-Poitiers [17], although the model is directly applicable for simulation of a general fire spread situation.

2.1 Experimental and simulation domain

In the experiments, a primary fuel source – a gas burner – was located in a zone of fresh air underneath a hood. A fire plume is generated above the burner and reaches into the hood, generating a zone of vitiated gas and soot. The distance between the burner and the hood was set so that insufficient air is entrained into the plume, ensuring that sufficient excess fuel is transported into the upper layer for the gas mixture to be flammable. As a consequence, combustion at the interface between the smoke zone and the fresh air below is sustained. A photograph of the installation is given in Figure 1.
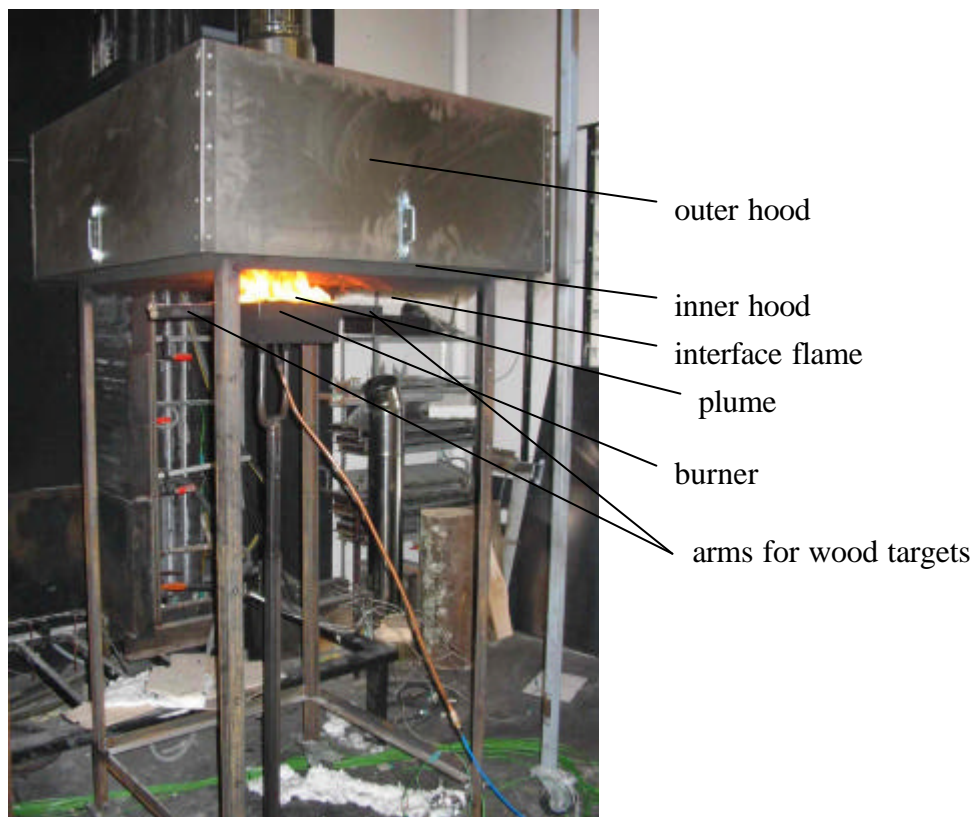

outer hood

inner hood

interface flame

plume

burner

arms for wood targets

Figure 1:   Photograph of the installation during one of the tests

The primary fuel source was a sand burner, which was supplied with 1.2 g/s of propane. This equates to a thermal power of approximately 55 kW. The tests were performed under a double-walled hood. The hood was designed so that smoke which spills out from underneath the inner hood is captured in the outer hood and led to a chimney, allowing measurements of the composition of the gases.

Three blocks from beech wood were placed on each arm radiating out from the sides of the burner. They were located at the same height as the surface of the burner, 0.17 m, 0.25 m and 0.33 m, respectively from the central vertical axis of the burner to the centre of the target. The

blocks were subject to heat transfer from the plume, the interface flame, the smoke layer, and the walls, such that the total heat flux was be sufficient for the objects to undergo unpiloted ignition. A more detailed description of the experimental tests is given in [17].
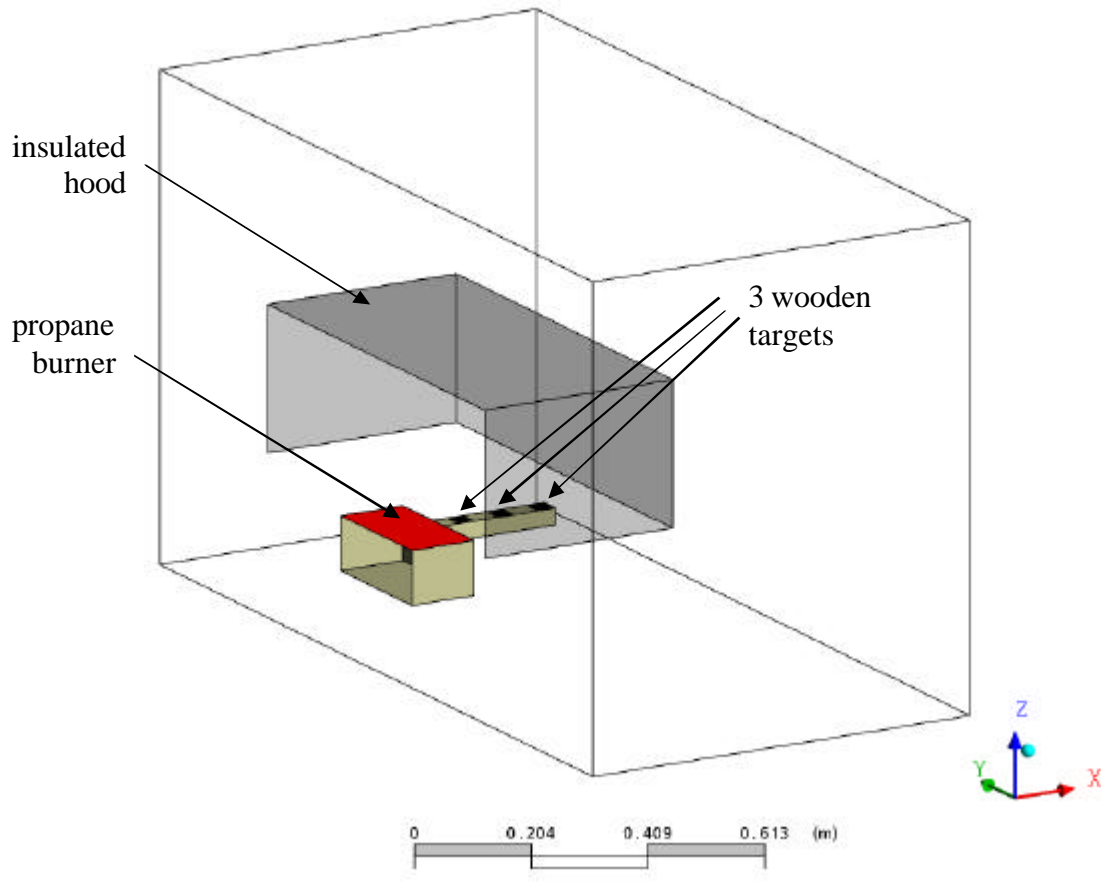

Figure 2:   Simulation domain of the CFD model

The numerical model includes most of the described components of the experimental setup. Only few simplifications are introduced:

- As symmetrical behaviour is expected, the simulation domain covers only half of the experimental installation (Fig. 3).
- The double hood arrangement is simplified and modelled as a single hood.
- The exhaust that collected smoke during the experiments is not taken into account.

2.2 Mathematical model

The model described transport of mass, momentum and energy in gaseous environment. It also includes combustion and thermal radiation. As the same time, degradation of wood samples is described with a pyrolysis model.

In the numerical model, pyrolysis of wood was approximated with a single step chemical reaction:

$$wood \rightarrow \nu_c \; char + volatiles \qquad (1)$$

The pyrolysis front is tracked with a progress variable that defines a solid mixture of virgin wood and char as proposed by Novozhilov et al. [18]. As well as a transport equation for the progress variable α, an energy transport equation for the solid mixture static enthalpy is solved. Material properties used in both tested models were taken from [12], [13] and [19].

Approximately 75 % of the wood mass is converted into volatiles, parts of which are combustible gases: H_2, CO, CH_4 and higher hydrocarbons:

$$volatiles \rightarrow \nu_g \; gas + residual \qquad (2)$$

Their total heat release rate is approximated with an equivalent mass flow of a single gas; in the present model, this is methane. By comparing the combustion heat of wood [20] with the combustion heat of methane, it is estimated that cumulative mass flow of combustible gases is approximately

$$m_{CH_4} \sim 0.361 \, m_{volatiles} , \qquad (3)$$

where $m_{volatiles} = 0.75 \, m_{wood}$. This mass flow is fuel source for the secondary fire, which appears above the wood samples during the simulations.

The mass, momentum and energy transport processes in gaseous environment are modelled using available functionality in the standard ANSYS CFX code. Flow dynamics and heat transfer are modelled using fully compressible formulation of Navier-Stokes equations. The flow's turbulence is described with the Shear Stress Transport (SST) model [21], which is a combination of the k-ε model and the k-ω model

The dominant heat transfer mechanism in the presented case is thermal radiation. In order to determine the radiation sources due to volumetric absorption in each control volume as well as the radiation heat fluxes on the walls, the radiation transport is modelled using the Discrete Transport model [22].

The gaseous mixture contains components of air (N_2 and O_2), propane (C_3H_8) as a source of primary fire, methane (CH_4) that models the gaseous phase of wood volatiles, and combustion products (CO_2 and H_2O). For the transport of C_3H_8, CH_4, O_2, CO_2 and H_2O, separate mass fraction transport equations are solved. The chemical reaction between the fuel of the primary fire (propane) and air is modelled as a single step chemical reaction. Also, the combustion of the gaseous phase of the pyrolysis volatiles (methane) is described with a single step chemical reaction. The reaction rate of these reactions is calculated with the eddy-dissipation model [23].

The hood plays an important role in the formation of the hot gaseous upper layer, and it is modelled with a single thin surface, at which a constant heat transfer coefficient is prescribed. The heat transfer coefficient is estimated from the thermal conductivity and thickness of the insulation layer between the inner and the outer hood.

6 Results and discussion

The geometrical and physical model of fire spread was solved using the ANSYS CFX software. To perform the transient numerical simulation for the presented geometrical arrangement, an unstructured computational mesh with approx. 0.3 million grid nodes was generated. A total of 12,500 of integration timesteps were needed to simulate 500 s of the flashover experiment.

At the start of the numerical simulation, a jet of propane is ignited and a mixture of hot gases that contains mostly combustion products starts to fill the hood above the burner. After the initial 10 s, the flame and the hot layer stabilize. They represent a constant source of thermal radiation that heats the upper surface of the wood samples.
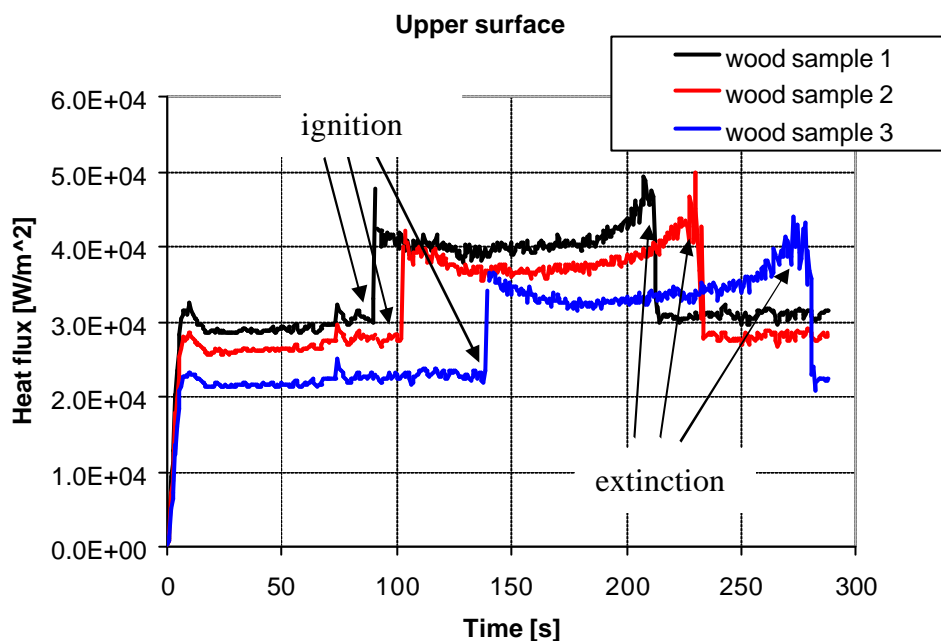

Figure 3: Wall convective and irradiation heat fluxes ($q_c + q_i$) variations

Figure 3 shows the sum of the convective and irradiation heat fluxes at the top surface of the wood samples. After the initial transient, when the hot layer stabilizes, the incoming heat flux reaches values between 20 and 30 kW/m^2, being higher for the wood surfaces that are closer to the burner. With the ignition of volatile pyrolysis products, the heat flux increases by approx. 12 kW/m^2. This rapid heat flux increase that marks the ignition event shows that the ignition time is different for each wood sample. Due to combustion of volatiles above the wood samples, the heat flux stays elevated until the wooden fuel is consumed. The time of extinction is different for each block, as well as the duration of wood degradation. After the extinction, the heat flux decreases to the values before the ignition.

Figures 4 present the temperature field above the three wood samples after the ignition of volatile pyrolysis products at 95 s, 120 s and 160 s, respectively. As such, Fig. 4a shows the fire above the wood sample 1, Fig. 4b above the samples 1 and 2, and Fig. 4c above all three samples. The presented temperature field also clearly shows the separation of the flow onto the hot upper layer, where the hood accumulates combustion products, and the colder lower layer with entrainment of the surrounding air.
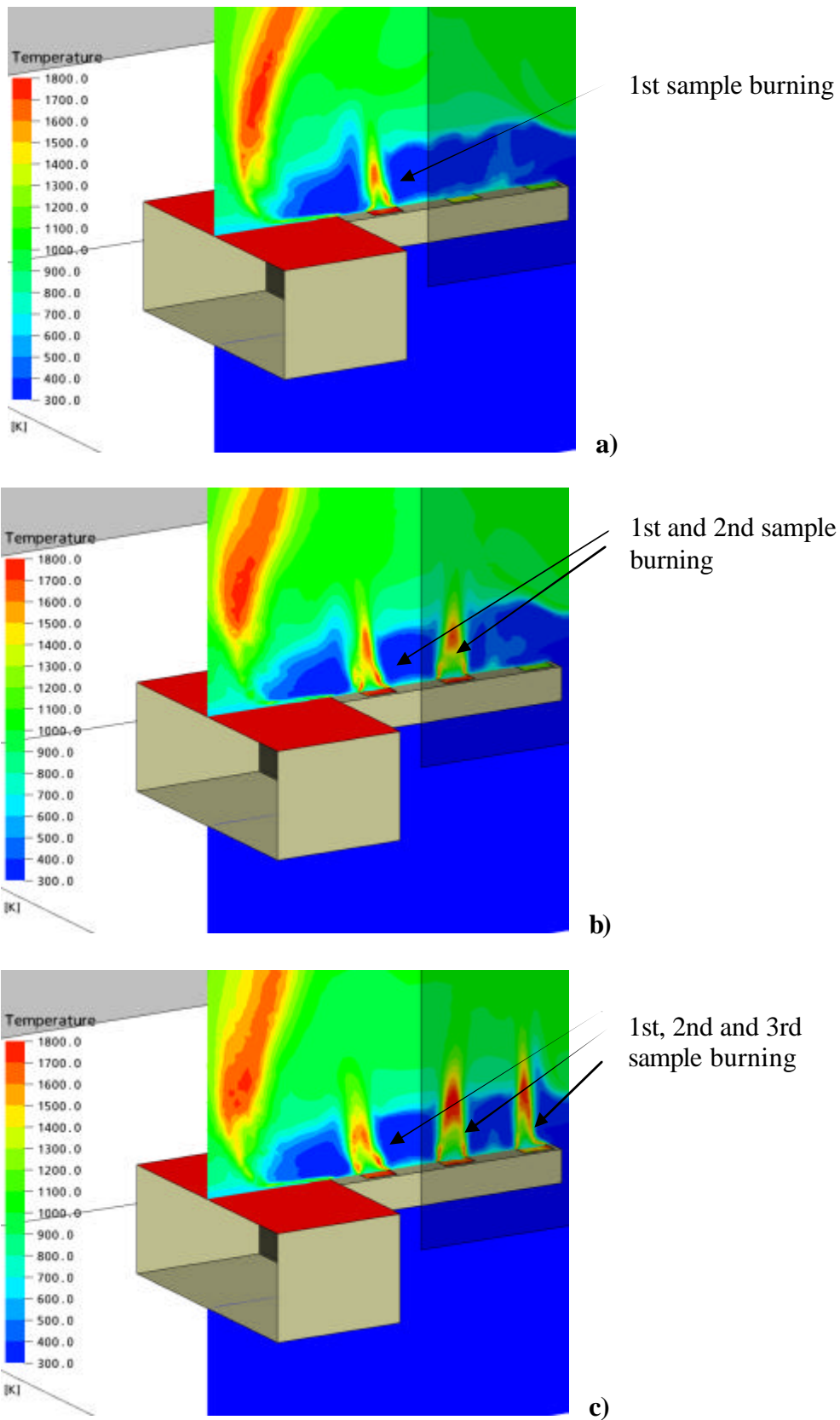
Figure 4: Temperature field cross section: **a)** at 95 s - after ignition of the first wood sample, **b)** at 120 s - after ignition of the second wood sample, and **c)** at 160 s - after ignition of the third wood sample

The temperature was recorded at the upper surface of the wood samples, which is exposed to convective and irradiation heat flux, and at the lower surface, which is embedded in the insulation.

Figure 5a shows the temperature at the upper surface of all the three wood samples. Initially, the temperature increases in a slightly parabolic manner followed by a rapid increase due to the ignition of pyrolysis volatiles. After the ignition event, the temperature increases with the same rate as before the ignition. The end of wood degradation is marked with a rapid acceleration of pyrolysis as the wood samples become practically isothermal.
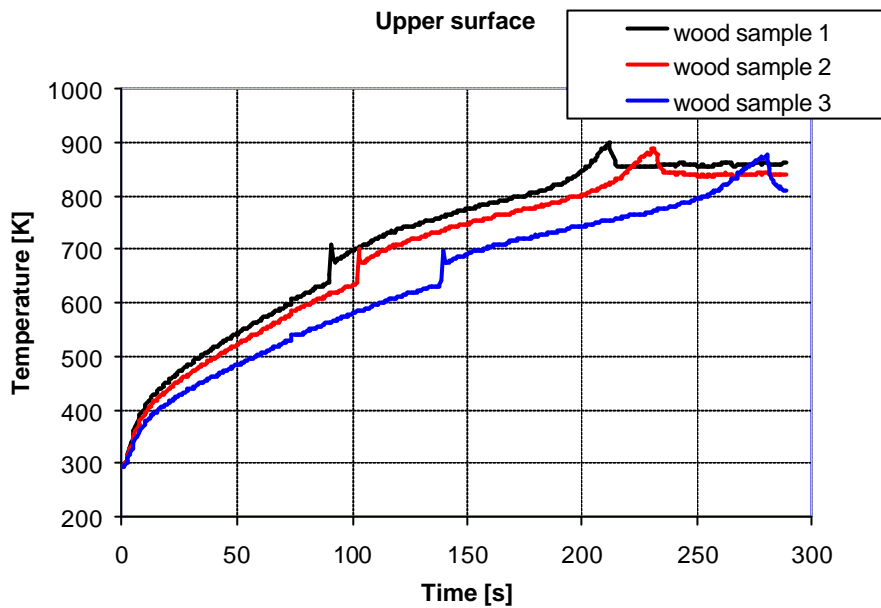
a)
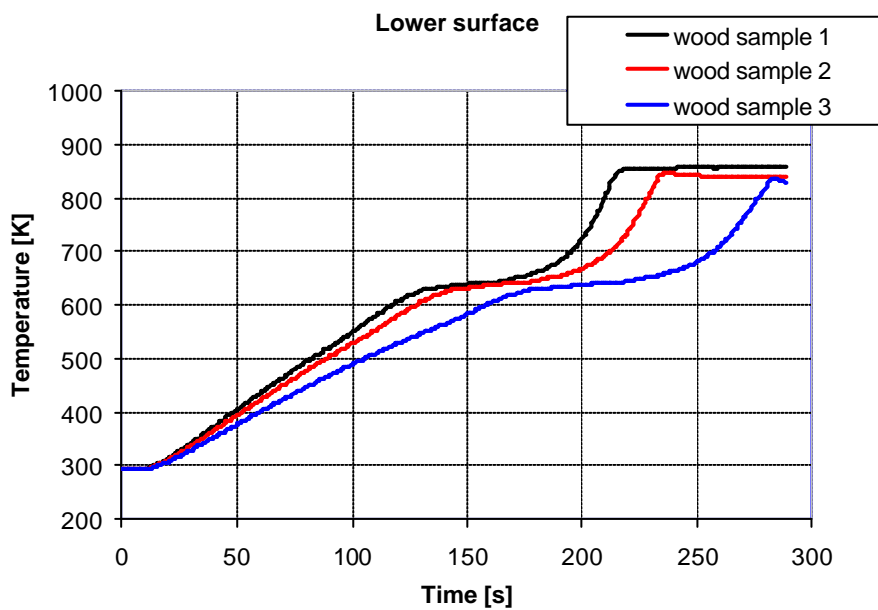
b)

Figure 5: Temperature variations: **a)** at the upper surface, **b)** at the lower surface of the wood samples

The temperature at the lower surface increases at the same rate as at the upper surface (Figure 5b), but with a certain delay as a result of energy diffusion from the upper to the lower surface. The reaction energy of pyrolysis is negative. Therefore, after the ignition the temperature increase stops temporarily and even decreases slightly.

The period of wood degradation is different for each sample as it depends on the amount of received heat. Figure 3 showed that the incoming (convective and irradiation) heat flux is the highest on the wood sample that is the closest to the propane burner. To validate the simulation approach, the calculated ignition time was compared to the experimental observations [17]. The ignition times for five experimental tests and for the numerical simulation are given in Table 4.

Table 1: Comparison of ignition times

	Ignition times [s]		
	Sample 1 (170 mm)	Sample 2 (250 mm)	Sample 3 (330 mm)
Test A	107	244	361
Test B	121	166	293
Test C	94	204	282
Test D	95	154	269
Test E	138	222	327
Numerical simulation	91	103	140

The ignition time for the first wood sample, which is the nearest to the propane burner, shows a good agreement with the experimentally obtained values, whereas the ignition times for the second and the third sample show increasing differences.

During the experiments, temperature was recorded at the upper surfaces of the three wood samples. Figures 6 show comparison of calculated and measured temperature variations. The calculated temperature variation for the first wood sample (Fig, 6a) matches well the experimentally obtained values over the whole simulation time. However, the time variations of the temperature for the second and the third wood sample (Figs. 6b and 6c) show that the model overpredicts the temperature from the beginning of the simulation, which leads to the premature ignition of the wood samples.
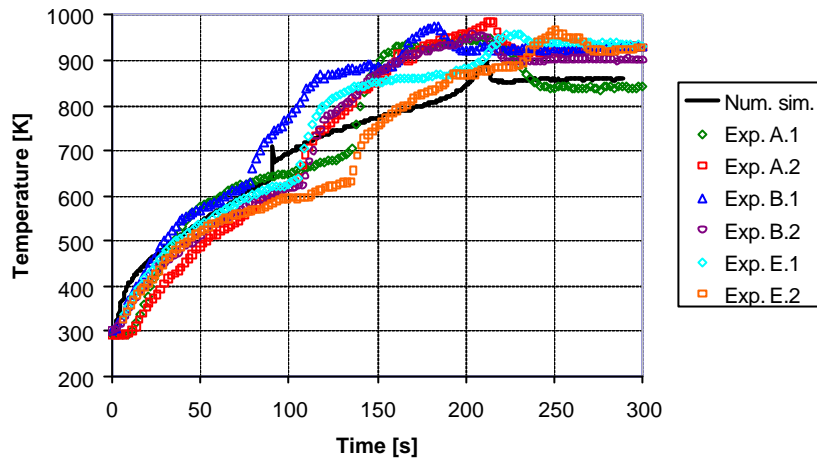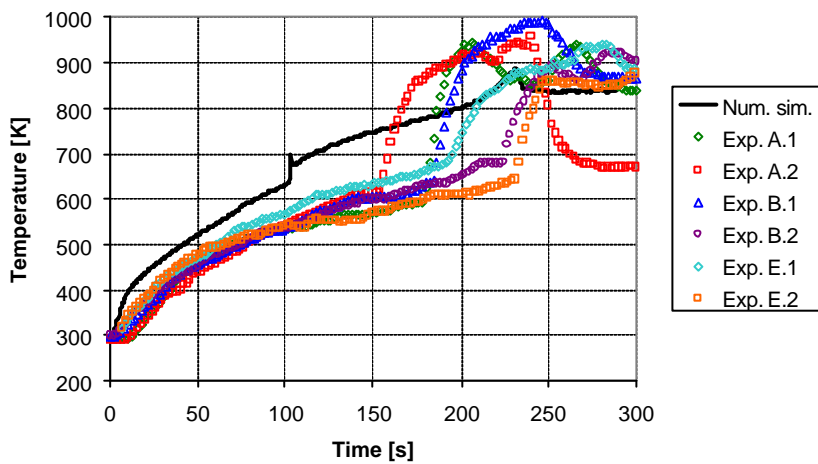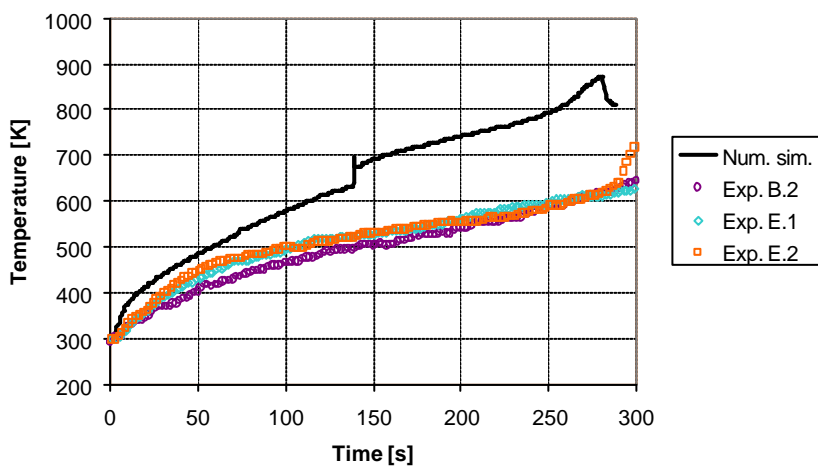
Figure 6: Comparison of temperature variations at the upper surface of the wood samples: **a)** sample 1, **b)** sample 2, and **c)** sample 3

In our opinion, the discrepancy between the experimentally obtained and the simulated ignition times, especially for the third wood sample, is a consequence of modelling simplifications. Reducing the model complexity by describing the double-walled hood as a single wall with a heat transfer coefficient, inadequately models the heat transfer from the hot layer of gases to the environment. As a result, the calculated temperature of the hot gas layer is overestimated and, therefore, emits stronger thermal radiation.

7 Conclusions

For prediction of the flashover phenomena, a CFD model was built that closely follows the experimental setup at CNRS-ENSMA-Poitiers [17]. The primary source of fire was a propane burner, and square beech wood samples were used as fire spread targets. Exposed to the heat fluxes from the fire, the temperature in the wood samples rises; at some stage the wood undergoes thermal decomposition and pyrolysis. In the numerical model, the pyrolysis front was tracked with a progress variable that defines a solid mixture of virgin wood and char. The total heat release rate of wood volatiles was approximated with an equivalent mass flow of methane.

To verify the developed model, a numerical simulation of the flashover experiment was performed. The collected results gave a qualitative insight into the radiation-induced ignition and fire spread over solid surfaces. It also helped us to identify important parameters of the flashover phenomenon and to evaluate their influence. The comparison of the calculated and the experimentally measured ignition times shows a good agreement that is well inside experimental variability for the first wood sample (0.17 m). For the second and the third wood sample, the calculated ignition time is shorter than the experimentally observed. Also, the calculated temperature's temporal variation at the upper surface of the first wood sample matches the experimental data well, but the differences are larger for the second and the third sample. We believe that these ignition time discrepancies are the consequence of the hood model simplification, as discussed above.

Although, the developed approach was validated using the specific experimental dataset [17], it can be also used in fire safety engineering as well as in other relevant industrial applications (e.g. for combustion of solid fuels). Alternative approaches to this modelling are also being explored at ANSYS Europe Ltd. It is reasonable to state, however, that this is a complex and challenging topic. It is important to emphasize that the presented computational model offers a prediction capability of a very hazardous phenomenon. This enables evaluation of engineering design and associated risk prior to construction and without performing full-scale tests.

References

1. Drysdale, D., An Introduction to Fire Dynamics, John Wiley & Sons, 1990.
2. Di Blasi, C., Physico-Chemical Processes Occurring Inside a Degrading Two-Dimensional Anisotropic Porous Medium, Int. J. Heat Mass Transfer, 1998, 41, pp. 4139-4150.
3. Karlsson, B., Quintiere, J.G., Enclosure Fire Dynamics, CRC Press, Boca Raton, USA, 2000.
4. Spearpoint, M.J., Mowrer, F.W., McGrattan, K.B., Simulation of a Compartment Flashover Fire Using Hand Calculations, Zone Models and a Field Model, Int. Conf. on Fire Research and Engineering (ICFRE3), Oct. 4-8, 1999, Chicago, IL, USA, Proceedings, pp. 3-14.
5. Peacock, R.D., Reneke, P.A., Bukowski, R.W., Babrauskas, V., Defining Flashover for Fire Hazard Calculations, Fire Safety J., 1999, Vol. 32, pp. 331-345.
6. Grosshandler, W. L., Role of Science and Engineering in Reducing Fire Risk, Fire Risk and Hazard Assessment Research Application Symposium, July 9-11, 2003, Baltimore, USA, Proceedings, pp. 1-21.
7. Babrauskas, V., Peacock, R. D., Reneke, P. A., Defining Flashover for Fire Hazard Calculations. Part 2, Fire Safety J., 2003, Vol. 38, pp. 613-622.
8. White, N., Dowling, V., Barnett, J., Full-Scale Fire Experiment on a Typical Passenger Train, The 8th Sym. of the Int. Assoc. of Fire Safety Science, Sept. 18–23, 2005, Beijing, China, Proceedings, Paper 112.
9. Luo, M., He, Y., Beck, V., Application of Field Model and Two-Zone Model to Flashover Fires in a Full-Scale Multi-Room Single Level Building, Fire Safety J., 1997, Vol. 29, pp. 1 -25.
10. Chow, W.K., Predictability of Flashover by Zone Models, J. Fire Sciences, 1998, Vol. 16, No. 5, pp. 335-350.
11. Novozhilov, V., Flashover Control under Fire Suppression Conditions, Fire Safety J., 2001, Vol. 36, pp. 641-660.
12. Atreya, A., Pyrolysis, Ignition and Fire Spread on Horizontal Surfaces of Wood, Ph.D. Thesis, 1983, Harvard University, Cambridge, MA, USA.
13. Fredland, B., A Model for Heat and Mass Transfer in Timber Structures During Fire: A Theoretical, Numerical and Experimental Study, Ph.D. Thesis, 1988, Dept. of Fire Safety Engineering, Lund Institute of Science and Technology, Sweden.
14. Aksit, M., Moss, J.B., Rubini, P.A., Field Modelling of Surface Flame Spread over Charring Materials, Interflam '01, Sept. 17-19, 2001, Edinburgh, Scotland, Proceedings, pp. 1459-1464.
15. Madrzykowski, D., Bryner, N.P., Grosshandler, W.L., Stroup, D.W., Fire Spread Through a Room With Polyurethane Foam Covered Walls, Interflam '04, July 5-7, 2004, Edinburgh, Scotland, Proceedings, Vol. 2, pp. 1127-1138.
16. Edwards, J.C., Hwang, C.C., CFD Modeling of Fire Spread Along Combustibles in a Mine Entry, The 2006 SME Annual Meeting and Exhibit, 2006, March 27-29, St. Louis, Missouri, USA, Proceedigs, No. 06-027.
17. Pearson, A., Most, J.-M., Data on Flashover, Report, June 2006, CNRS, University of Poitiers, ENSMA, Laboratoire de Combustion et de Détonique, UPR 9028, Poitiers, France.
18. Novozhilov, V., Moghtaderi, B., Fletcher, D.F., Kent, J.H., Computational Fluid Dynamics Modelling of Wood Combustion, Fire Safety J., 1996, 27, pp. 69-84.
19. Di Blasi, C., Branca, C., Santoro, A., Hernandez, E.G., Pyrolytic Behavior and Products

of Some Wood Varieties, Comb. and Flame, 2001, 124, pp. 165-177.

20. Robert, A.F., Calorific Values of Partially Decomposed Wood Samples, Combustion and Flame, 1964, Vol. 8, pp. 345-346.
21. Menter, F.R., Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications, AIAA Journal, 1994, Vol. 32, No. 8.
22. Shah, N.G., New Method of Computation of Radiant Heat Transfer in Combustion Chambers, Ph.D. thesis, 1979, Imperial College, London, United Kingdom.
23. Magnussen, B. F., Hjertager, B. H., On Mathematical Modeling of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion, The 16th Symp. (Int.) on Combustion, 1976, Proceedings, pp. 719–729.