CHEMICAL ENGINEERING TRANSACTIONS

VOL. 36, 2014

A publication of
AIDIC
The Italian Association
of Chemical Engineering
www.aidic.it/cet

Guest Editors: Valerio Cozzani, Eddy de Rademaeker
Copyright © 2014, AIDIC Servizi S.r.l.,
ISBN 978-88-95608-27-3; **ISSN** 2283-9216

DOI: 10.3303/CET1436042

A Critical Comparison Between CFD and Zone Models for the Consequence Analysis of Fires in Congested Environments

Silvia Tavelli, Renato Rota, Marco Derudi*

Politecnico di Milano, Dip. di Chimica, Materiali ed Ingegneria Chimica “G.Natta”, piazza Leonardo da Vinci 32, 20133, Milano, Italy
marco.derudi@polimi.it

Fire represents a very complex phenomenon, as the severity of the consequences associated with its evolution depends not only on the fire source itself, but also on the configuration of the system in which it develops, that is whether the source term is placed in an unconfined environment or it is confined in a room or in an elongated space such as a tunnel, either empty or filled with vehicles. Numerical modeling approaches allow to obtain cost-effective and reproducible information, with a degree of confidence dependent on the quality of the input information and the assumptions made by the model itself. In this study two different modeling approaches have been compared, a computational fluid dynamics code and a zone model in order to test their effectiveness and their limitations in predicting different congested fire scenarios.

A series of tests has been performed using a single-room configuration that is the most classical application of zone models, comparing the models performances with experimental results and highlighting the scarce sensitivity of zone models to some aspects, such as the source term position. A simple tunnel test has then been reproduced, testing the different models capabilities of identifying general trends and fundamental parameters, such as the critical ventilation velocity and the temperature profiles along the elongated geometry. Moreover, both models have been used to predict the effect of the presence of obstacles in the computational domain, showing that zone model cannot account for their presence.

The simulation results highlight that, although the simulation times and the expertise required in order to implement and process the information are much lower, zone models can be useful only in very simple configurations; moreover, they can be extended with some limitation to complex room fires and cannot be relied upon for tunnel fires analysis, where fundamental aspect such as the critical ventilation velocities are not correctly predicted.

1. Fire in congested environments and safety issues

Fire represents a complex phenomenon involving different interacting physical and chemical processes at different scales, such as combustion, radiation, transport phenomena under turbulent conditions. Its evolution and the severity of the consequences experienced depend on the boundary conditions around the fire, that is whether the source term is placed in an unconfined environment or it is confined in a room or in an elongated space, such as a duct or a tunnel.

Accidental fires often develop in congested environments such as production areas, warehouses, urban areas (Pontiggia et al., 2011) or tunnels (Beard and Carvel, 2005; Tavelli et al., 2013a), and if not properly handled can lead to severe consequences in terms of loss of lives, structural damages and economic losses (Kang, 2010; Tavelli et al., 2013b). Given the complexity of the scenarios, the role of reliable simulation tools is becoming fundamental for design purposes of new tunnels, for improvement of existing ones and for buildings with peculiar structural characteristics, according to the evolution of standards and guidelines (2004/54/EC). Numerical modeling approaches allow to obtain cost-effective and reproducible information, with a degree of confidence dependent on the quality of the input information and the

Please cite this article as: Tavelli S., Rota R., Derudi M., 2014, A critical comparison between cfd and zone models for the consequence analysis of fires in congested environments, Chemical Engineering Transactions, 36, 247-252 DOI: 10.3303/CET1436042

assumptions made by the model itself. In this work we compare different simulation approaches in order to test their capability and their limitations in predicting different congested fire scenarios: the computational fluid dynamics code FDS and the zone model CFAST, both open source codes – made by the NIST - specific for fires modeling.

2. Numerical approaches

2.1 CFD

Three-dimensional computational fluid dynamics (CFD) codes implement the fundamental equations of fluid dynamics over complex domains. They solve numerically the equation of continuity, the conservation of momentum and energy as a system of partial differential equations, relying on few submodels in order to take into consideration aspects such as turbulence description, reaction kinetics, radiation transport and pyrolysis.

The Fire Dynamic Simulator (FDS) is an open source CFD model for fire driven fluid flows. The code solves numerically a form of the Navier-Stokes equations appropriate for low speed, thermally-driven flows, coupling them with submodels which account for approximate descriptions of the combustion reactions and the soot formation. Turbulence is described with the Smagorinsky form of Large Eddy simulation (LES): the eddies that account for most of the mixing are large enough to be calculated with reasonable accuracy from the equations of fluid dynamics, while the small scale eddy motions are accounted for with a sub grid scale model (McGrattan et al., 2010a). The combustion model implemented in FDS5, the version used in this work, relies on the concept of mixture fraction, a conserved scalar quantity defined as the fraction of gas that originates as fuel in the flow field in a specific position in the domain. Thermal radiation is computed using a finite volume technique on the same grid as the flow solver; the solution of the radiation transport equation for the grey gas is obtained by subdividing the radiation spectrum into a relatively small number of bands. The partial derivatives of the conservation equations are approximated as finite differences, and the solution is updated on a three dimensional, rectilinear grid to which all the geometric features of the scenario must conform.

Due to the accuracy of their phenomenological description and the potential variety of configurations and boundary conditions they can describe, CFD models allow the user to model the interactions which occur simultaneously in a fire accident, helping to assess the influence of different parameters on the event evolution. It is possible to represent the geometry of the system, define boundary conditions as well as to add sprinklers and smoke detectors to the simulation, simply by describing their position and describing their properties in the Cartesian coordinate system. All the geometrical entities are divided into rectilinear grid cells, and their description is achieved by defining their coordinates in the computational domain in a text input file; in case of more complex geometries with curved walls, the user can draw the geometry of the system into the 3D environment of the software platform Blender. Blender is an open source suite for the creation of 3D contents; a specific add-on called BlenderFDS (Gissi, 2012) can be used to convert the geometry into an FDS input file, making the geometrical characterization of the computational domain less cumbersome.

Since its release, FDS has been subject to a series of validation tests over different configurations, thus it is considered appropriate for complex geometries descriptions (Wu and Bakar, 2000; Yeoh and Yuen, 2008). However, the CFD approach requires extensive computational time and resources in order to perform effectively.

2.2 Zone Models

Zone models rely on the assumption that a volume can be vertically subdivided into zones, perfectly mixed and with homogeneous properties in terms of temperature and composition: a hot layer with combustion products, located near the ceiling, and a cold layer with fresh clean air at the bottom, separated by a moving interface. The properties (and the layer height) can vary over time and are identified when solving global conservation equations.

CFAST is a two zone fire model used to calculate the smoke dispersion, the fire gases dynamics and the temperature throughout compartments of a constructed facility over time; each compartment is divided into two gas layers. The fundamental equations (conservation of mass and energy over the layers, ideal gas law and relations for density and internal energy) are implemented as system of ordinary differential equations (ODEs), which are solved to give the values of pressure, layer heights and temperatures over time. A series of algorithms allow to compute the mass and enthalpy source terms required by the ODEs (McGrattan et al., 2010a).

Zone models rely on very strong simplifications as well as on experimental information, and their application is limited to the geometry characteristics over which the model was tested and validated. For instance, the complex three dimensional flow of gases and hot products rising from a fire located inside a tunnel cannot be represented with a two-zone approach with a single compartment to represent the whole tunnel volume. The division of the compartments in two horizontal layers spanning over the entire compartment area, would not take into consideration the destratification of smoke downstream from a fire or the backlayering of the smoke upstream of the source (Tavelli et al., 2013a).

Zone models are commonly used in risk analysis of rooms and simple buildings (Floyd, 2002). CFAST has an interface which guides the user into the construction of the scenario, allowing to describe the geometry of the system, the openings of the compartment to the outside, the mechanical ventilation flows, as well as the fire source term. A database of both fire sources (and burning rate curves) and building materials thermal properties, which can be modified by the user, are also provided in the software. Moreover, the simulation times are in the order of seconds.

3. Results and Discussion

In this study, different scenarios have been analysed. As previously mentioned, the aim was to compare the performances of CFD and zone models for different fires in congested environments, such as room fires, for which the zone models have been extensively validated (McGrattan et al, 2010a) and tunnel fires, which represent a more complex scenario that in principle is difficult to model with zone assumptions. After a validation against experimental data, the models sensitivity to some aspects, such as the presence of obstacles and the effect of a mechanical ventilation have been further analysed.

3.1 Room Fire Analysis

The first scenario reproduces a series of tests performed by Steckler et al. (1982) aimed at analysing the stratification and the temperature layers achieved in a room in steady state conditions. The domain was a 2.8 x 2.8 x 2.18 m^3 test room with adiabatic walls, and the series of tests recorded temperatures, velocities and layer height inside the compartment when varying the door (or window) opening as well as the location and the heat release rate (HRR) of the burner inside the room.

The set up of the system was straightforward in CFAST, while in FDS the system had to be modelled by dividing the domain into a mesh of 387000 cells; the number of grid cells over the fire source should be enough to correctly represent the scenario without grid dependence effects (McGrattan et al., 2010b). The average simulation times were of a few seconds with CFAST, about 45 h with FDS.

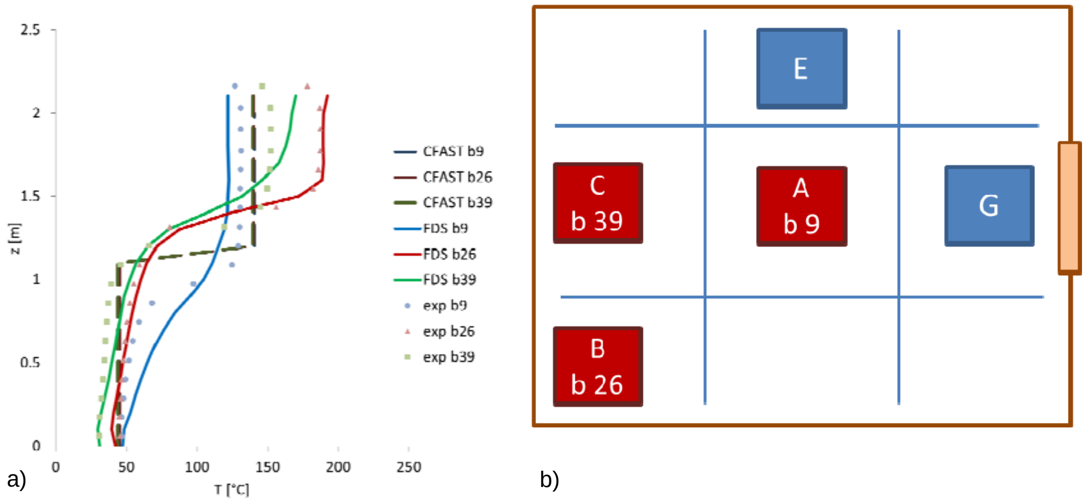

Figure 1 a) Temperature profiles obtained in the room corner for different burner locations: comparison among CFD (continuous lines), zone models (dashed lines) and experimental results (symbols) when varying the burner location (positions A, B, C); b) burners and obstacles locations inside the room

Both models were able to qualitatively identify the gas stratification and flow patterns generated by the fire source, as well as the effect of the different HRR on the layer temperatures: the predictions when doubling and halving the fire HRR have a difference of 8-24 % from the experimental results when using the zone model, of 1-6 % with the CFD. The zone model description of the compartment, however, does not allow to account for the differences of a variation in the location of the fire source, as shown in Figure 1a. The subdivision of the room into several fictitious compartments (mimicking the cells subdivision in CFD) has been then performed in CFAST, in order to try to overcome the limitation of the single zone approach; a multi-compartment environment was realized by dividing the domain and locating horizontal vents between the different compartments (thin lines in Figure 1b). Tests were conducted with 3 (along either axis) and 9 compartments, respectively. The results, in terms of relative difference between simulated and experimental measurements, showed that the subdivision allow to describe the effect of the variation of the burner position. Moreover, an increase in the number of compartments improves significantly the predictions for the non-simmetricl test cases (burners B and C), which for the top layer can reach an accuracy of more than 95 %, as shown in Figure 2. On the other hand, CFAST predictions for the bottom layer are less accurate and can worsen with an increase of the number of compartments.

However, in spite of the CFAST predictions can be improved by increasing the number of compartments, in case no experimental or CFD results are available it is not possible to identify which subdivision is accurate enough. Moreover, the zone model is intrinsically not able to reproduce the influence of the presence of obstacles inside the room.

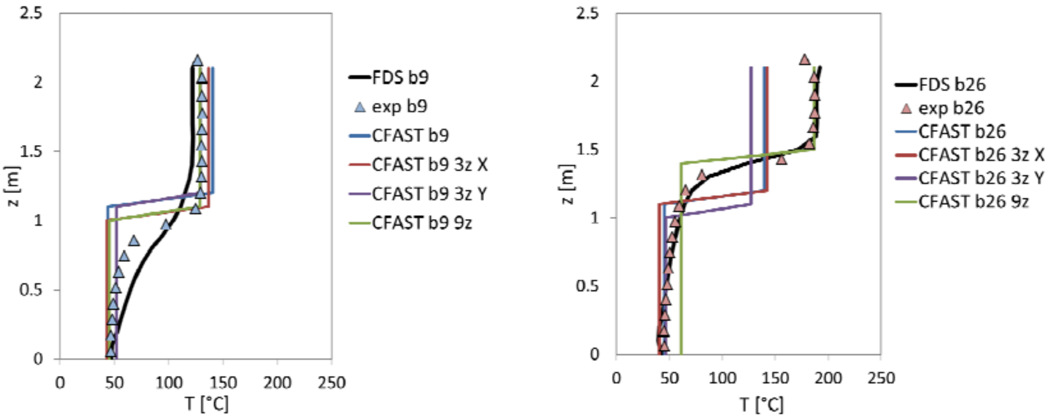

Figure 2 Temperature profiles obtained in the room corner for burner in position A (left) and B (right), when varying the compartment subdivision in CFAST (3 zones subdividing the room along the X or Y axis, and 9 zones, as shown in figure 1b). Both experimental and FDS results are also included for comparison

3.2 Tunnel Fire Analysis

The tunnel fire analysis relied on the tests performed by Apte et al. (1991) to study the smoke flow under different ventilation conditions in a reproduction of a mine tunnel (130 m long, 5.4 m wide, 2.4 m high) with rectangular section. The experimental analysis included three case studies (with longitudinal ventilation velocities of 0.5 m/s, 0.85 m/s and 2.0 m/s, respectively) which have been modelled in FDS on a 400000 cells computational domain, with 0.1x0.1x0.1 m^3 cells over the fire source, coherently with the criterion developed by Quintiere (Ma and Quintiere, 2003) and stretched to the tunnel ends. The fire source has been modelled as a burner with assigned HRR (derived from the experimental tests), activated only after a time span which allowed the development of a fully turbulent ventilation flow inside the tunnel.

The simulation in CFAST has been realized by dividing the domain into a number of compartments along the tunnel axis, in order to try and overcome the limitations of the single-compartment approach, which does not allow to describe the destratification along the tunnel length. As Figure 3a shows, when varying the number of compartments in which the tunnel is divided, the results can vary considerably. In the following, the maximum number of compartments (that is, 19 zones) will be used.

As an example, the hot gas layer position as well as the temperature distribution predicted by CFAST and FDS are reported in Figure 4 for one of the investigated cases. The limitation of CFAST are evident from this figure, resulting in wide different temperature values in each lower compartment. Moreover, CFAST cannot account for the effect of obstacles inside a compartment, as evidenced by Figure 5, where temperatures predicted by FDS and CFAST are reported for situations involving large obstacles.

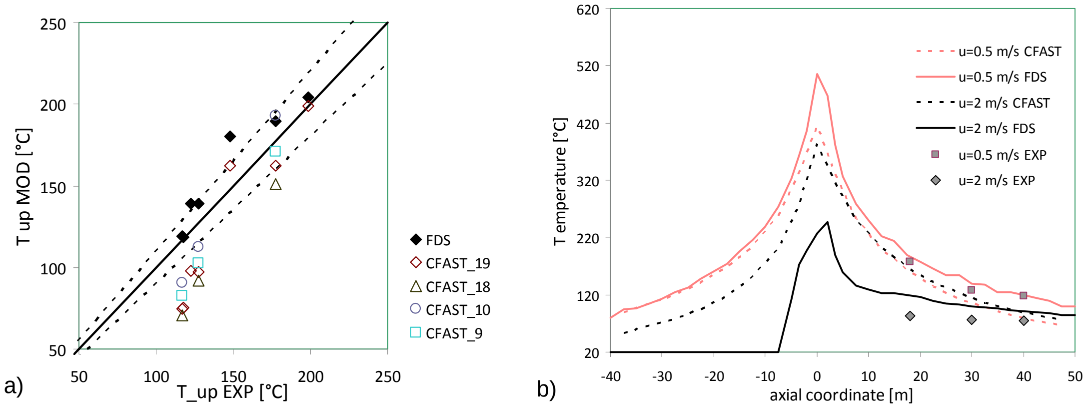

Figure 3 a) Experimental measurements (test cases with u=0.5 m/s and u=0.85 m/s) compared with FDS and CFAST predictions with a different number of compartments (dashed lines represent +/- 10 % differences); b) Effect of the variation of the longitudinal ventilation on the predicted temperature profiles

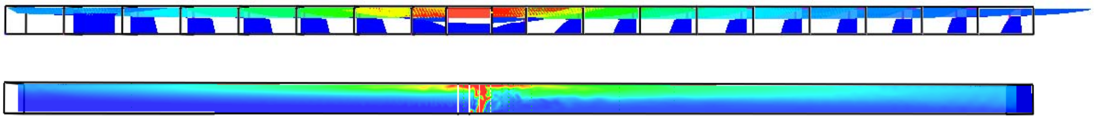

Figure 4 Temperature distribution inside the tunnel predicted by CFAST (top), with the 19 zone configuration, and FDS (bottom) for the test case with u=0.85 m/s

Even when taking into consideration the variations in the ventilation flow rates by modifying the horizontal ventilation links among the compartments, the temperatures predicted by CFAST were always similar downstream of the fire. On the other hand, FDS highlights a strong sensitivity to both the presence and the position of the obstacles.

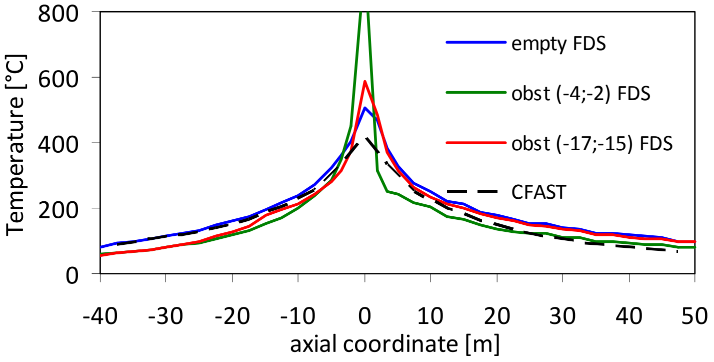

Figure 5: Effect of the presence of obstacles along the tunnel on the upper gas layer temperature (u=0.5 m/s) for FDS and CFAST (which shows identical results in all three tests). An obstacle of 2x2x2 m^3 was located inside the tunnel, along the axis, either 15 or 2 m upstream from the fire source, compared with the reference case of empty tunnel

The comparisons among the different test cases simulated with the two different models show that, although CFAST achieves reasonable agreement with low ventilation velocities, when the smoke layer extends over the entire tunnel length it cannot represent the effect of the increase in ventilation velocity, when the smoke backlayering length extended over a short length of tunnel (test with u=2 m/s, Figure 3b). This is confirmed by an analysis aimed at identifying the critical ventilation velocity, u_{cr}, along the tunnel.

While FDS simulations indicate a u_{cr} of about 2.5 m/s, with no difference when assigning the ventilation input from either tunnel ends, the simulation results in CFAST vary considerably and are unrealistically high (u_{cr}=13.8 m/s when imposing an exhaust ventilation, u_{cr}=9.9 m/s when the air is supplied from the upper end of the tunnel). These results, in particular, lead to the conclusion that CFAST cannot be relied upon when performing tunnel fire analysis.

4. Conclusions

Given the severity of the fire accidents in congested environments, the study of this phenomena with numerical tools is becoming widespread. Given the differences in terms of computational resources, simulation times and expertise required in order to perform an analysis with simulation tools of different levels of approximation, the results obtained have been compared by simulating the same experimental tests with CFD and zone models. Moreover, a sensitivity analysis allowed to evaluate the influence on the smoke and fire dynamics of several parameters such as the fire position, the fire heat release rate, the ventilation characteristics and the presence of obstacles.

While both models can provide reasonable results when modelling empty room fires, the intrinsic simplifications of the zone models make them unreliable when performing tunnel fire simulations or when large obstacles are present inside the computational domain. These results underline the importance of CFD models (with respect to zone models) as fundamental tools to increase the understanding of fire accidents, and their potential role in identifying or designing adequate prevention and protection systems.

References

Apte V.B., Green A.R., Kent J.H., 1991, Pool fire plume flow in a large-scale wind tunnel, Fire Safety Science Proceeding of the 3rd International Symposium, 425-434.

Beard A., Carvel R., 2005, The handbook of tunnel fire safety, Thomas Telford Publishing, London, UK.

Directive 2004/54/EC of the European Parliament on “Minimum safety requirements for tunnels in the Trans-European Road Network”, Annex 1, 29 April 2004.

Gissi E., 2012, Installing BlenderFDS <code.google.com/p/blenderFDS>, accessed 20.03.2012.

Floyd J.E., 2002, Comparison of CFAST and FDS for Fire Simulation with the HDR T51 and T52 Tests, NIST Report NISTIR 6866.

Kang K., 2010, Characteristic Length scale of critical ventilation velocity in tunnel smoke control, Tunneling and Underground Space Technology 25, 205-211.

McGrattan K., Hostikka S., Floyd J., Baum H., Rehm R., Mell W., McDermott R., 2010a, Fire Dynamics Simulator (Version 5). Technical Reference Guide, NIST Special Publication 1018-15, US Government Printing Office, Washington, USA

McGrattan K., McDermott R., Hostikka S., Floyd J., 2010b, Fire Dynamics Simulator (Version 5). User’s Guide, NIST Special Publication 1019-5, US Government Printing Office, Washington, USA.

Ma T.G., Quintiere J.G., 2003, Numerical simulation of axi-symmetric fire plumes: accuracy and limitations, Fire Safety Journal 38, 467-492.

Pontiggia M., Landucci G., Busini V., Derudi M., Alba M., Scaioni M., Bonvicini S., Cozzani V., Rota R., 2011, CFD model simulation of LPG dispersion in urban areas, Atmospheric Environ. 45, 3913-3923.

Steckler K.D, Quintiere J.G., Rinkinen W.J., 1982, Flow induced by fire in a compartment, National Bureau of Standards, Washington, USA.

Tavelli S., Derudi M., Cuoci A., Frassoldati A., 2013a, Numerical analysis of pool fire consequences in confined environments, Chemical Engineering Transactions 31, 127-132.

Tavelli S., Rota R., Derudi M., 2013b, Validation of FDS for predicting fire behavior in congested tunnels, submitted to Fire Safety Journal.

Yeoh G.H., Yuen K.K., 2008, Computational Fluid Dynamics in fire engineering: theory, modeling and practice, Elsevier, NL.

Wu Y., Bakar M.Z.A., 2000, Control of smoke flow in tunnel fires using longitudinal ventilation systems – A study of the critical velocity, Fire Safety Journal 35, 363-390.