

A Field Modeling Approach to Prediction of Hot Gas Movement Induced by Marine Compartment Fires

Changhong Hu

Research Institute for Applied Mechanics, Kyushu University, Fukuoka, Japan

Nobuyoshi Fukuchi

Graduate School of Engineering, Kyushu University, Fukuoka, Japan

ABSTRACT

Predicting smoke flow movements in marine and offshore compartment fires is still a challenging endeavor for fire safety researchers. The research described in this paper is an effort to apply the computational fluid dynamics (CFD) model to numerically simulate such smoke flows. The present computational model involves a CFD code to solve 3-dimensional turbulent buoyancy-driven flows using a set of low-Mach-number approximated governing equations, a turbulence model based on large eddy simulation, and a combustion model to calculate the heat release rate from fires. A pool fire is simulated and the results are compared to the measured data of a laboratory experiment. A simulation of an engine room fire on a coastal LPG ship is also presented as an example of applications.

INTRODUCTION

For most of recorded maritime history, fire has been a major cause of loss of ships and offshore platforms. To design a fire safety system for a ship or an offshore platform is then one of the most important problems for the functional designers. In the design process of fire detection and extinguishing systems, knowledge of data about heat transfer and smoke movement after the breakout of a fire is often required.

Numerous studies have attempted to model fire phenomena; the newest review is provided by Tieszen (2001). The zone models, in which room-averaged quantities are predicted for multi-room problems, have been used extensively for engineering applications. Recently, field models, which make the simulation possible in much finer spatial and temporal resolution, have been used in the study of fires. The research described in this paper concerns a field modeling approach under development.

A distinguishing feature of the fire problem is that the temperature difference as well as the density variation are very large, while the speed of the smoke flow induced by the heat release is much slower than that of the propagation of acoustic waves. Such flows are thus usually classified as low-Mach-number compressible flows. The generally used Boussinesq approximation, in which the density is assumed constant except for what appears in the buoyancy term of momentum equations, is limited to a very small temperature difference and is not suitable for the fire problem (Gray and Giorgini, 1976). This paper uses a set of low-Mach-number approximated equations. As the smoke flows are usually turbulent, models are required to account for the effect of unresolved small-scale movements. The most commonly used

turbulence model for the simulation of practical fire problems is the $k-\varepsilon$ model. The weak point of the model is its low temporal resolution due to the time-average process. The large eddy simulation (LES) approach, in which spatial filters are used to derive the governing equations for resolved variables, is a very promising model that can obtain time-accurate field information and overcome the weak point of the $k-\varepsilon$ model. The usefulness of the LES method on fire simulations has been demonstrated by McGrattan et al. (1998). LES is chosen as the turbulence model for the present computation.

In addition to the effort of improving the precision of the solution of the discretized control equations for fire-induced turbulent flows, another important and more complicated endeavor is to determine the heat release rate of the fire, which involves chemical and physical processes such as ignition, combustion and extinction. To avoid such complications, usually the heat release rate is given by empirical data as a user input in many computational fluid dynamics (CFD) fire models. In order for the simulation to be closer to a real fire situation, the present CFD method uses a combustion model based on the calculation of gaseous fuel and air mixing so as to calculate the heat release rate as well as the flame's position.

This paper first describes the methodology, which involves an explicit calculation of large eddy structures in the fire plume using a set of spatial filtered low-Mach-number approximated Navier-Stokes equations, a turbulence model based on the Smagorinsky SGS model (Smagorinsky, 1963) to account for the effect of sub-grid fluctuations, and a combustion model to determine the heat release rate. Two examples are then presented. The first is a 3-D numerical simulation of a pool fire. The computed temperatures within the fire plume are compared with experimental data to validate the precision of the method. The second example is a simulation of an engine room fire on a coastal LPG ship. The early stage of hot gas movements after the breakout of fire is simulated to demonstrate the suitability of the present CFD model for the fire problem in enclosed spaces with complicated geometries.

Received July 8, 2002; revised manuscript received by the editors October 9, 2003. The original version (prior to the final revised manuscript) was presented at the 12th International Offshore and Polar Engineering Conference (ISOPE-2002), Kyushu, Japan, May 26–31, 2002.

KEY WORDS: Computational fluid dynamics, field model, low-Mach-number approximation, large eddy simulation, combustion model, marine compartment fire.