Validation of the Isis-3D Computer Code for Simulating Large Pool Fires Under a Variety of Wind Conditions

The Isis-3D computational fluid dynamics/radiation heat transfer computer code was developed to simulate heat transfer from large fires to engulfed packages for transportation risk studies. These studies require accurate estimates of the total heat transfer to an object and the general characteristics of the object temperature distribution for a variety of fire environments. Since risk studies require multiple simulations, analysis tools must be rapid as well as accurate. In order to meet these needs Isis-3D employs fuel evaporation reaction rate and radiation heat transfer models that allow it to accurately model large-fire heat transfer even when relatively coarse computational grids are employed. Reaction rate and soot radiation model parameters in Isis-3D have been selected based on experimental data. In this work, Isis-3D calculations were performed to simulate the conditions of three experiments that measured the temperature response of a 4.66 m diameter culvert pipe located at the leeward edge of 18.9 m and 9.45 m diameter pool fires in crosswinds with average speeds of 2.0, 4.6, and 9.5 m/s. Isis-3D accurately calculated the time-dependent temperatures in all three experiments. Accelerated simulations were performed in which the pipe specific heat was reduced compared to the measured value by a factor of four. This artificially increased the speed at which the pipe temperature rose and allowed the simulated fire duration to be reduced by a factor of four. A 700 sec fire with moderately unsteady wind conditions was accurately simulated in 10 hours on a standard workstation. [DOI: 10.1115/1.1767173]

Introduction

Transportation and storage of high-level nuclear waste, military materials, and other hazardous substances involve risks from accidental or intentional insults. Risk assessment studies estimate the probability that severe events will take place and their likely consequence [1]. These studies rely on computer models to estimate the physical consequence of impact, fire, water immersion and other severe events. Since a wide variety of events must be considered, these models must be rapid as well as accurate. In the current work we are concerned with predicting the response of objects in large fires.

Specialized fire physics codes such as Kameleon from SINTEF, Vulcan from Sandia, and commercial computational fluid dynamics codes, such as CFX developed by AEA Harwell, and Fluent, are capable of calculating the flow, temperature and species fields within fires. They calculate fire behavior from first principles but may require massive amounts of run time on specialized computing platforms. These codes are also capable of calculating conjugate conduction and radiation within objects engulfed in fires. However, they employ the same grid structure for both the flowing and solid regions. If the solid regions incorporate small-scale structures that require small mesh dimensions, the computational time step becomes prohibitively small. These codes are therefore not well suited for the multiple simulations required in risk studies.

On the other hand, simple fire models that involve a specified fire temperature and effective fire emissivity have also been employed [2]. They are easily linked to finite element models of the hazardous material package, and the linked model produces results with relatively short computational turnaround times. However, these models generally do not include the effects that wind, other objects or enclosures play on the heat transfer from a fire to the package.

The Isis-3D computational fluid dynamics/radiation heat transfer computer code is under development as a tool for risk assessment and engineering level analysis. Its primary purpose is to provide reasonably accurate estimates of the total heat transfer to objects from large fires under a variety of circumstances, predict the general characteristics of the object temperature distribution, and use fairly short computer turnaround times. Isis-3D models liquid fuel evaporation, transport of fuel vapor, oxygen and other relevant species, reaction and heat release, and soot and intermediate species formation/destruction. It models diffuse radiation within the fire, and view factor radiation from the fire edge to nearby objects and the surroundings.

One-dimensional transient conduction modules are also embedded in Isis-3D. Either or both “ends” of each module are coupled to the flowing medium region. These modules allow the code to calculate the response of simple solid objects to the fire environment without affecting the computational fluid dynamics time step. The Container Analysis Fire Environment (CAFE) computer code is another version of Isis-3D that allows the flowing medium to be linked to a finite element program [3]. In CAFE the finite element program is used to calculate the detailed three-dimensional response of complex objects (such as a nuclear waste transport package) to a fire.

Isis-3D is a general-purpose three-dimensional computational fluid dynamics code that is capable of employing highly refined computational meshes. The radiation heat transfer and chemical reaction models embedded in the code are designed to enable it to give engineering-level accurate results for large-fire heat transfer even when relatively coarse computational grids are employed. Moderate-resolution Isis-3D simulations (less than 60,000 nodes)
are relatively fast running on desktop workstations and hence well suited for risk assessment studies and engineering level analyses. The reaction rate, soot production and radiation heat transfer models employed in Isis-3D use a number of parameters. The parameter values were selected based on experimental data from two large fire tests [4]. In the first test, time-dependent soot volume fraction and soot temperature measurements were performed in a 6 m by 6 m square pool fire under light wind conditions [5]. The second test measured local temperatures of a large culvert pipe suspended near the leeward edge of an 18.9 m diameter pool fire in a 9.5 m/s crosswind [6]. The measured wind speed and direction exhibited rapid fluctuation during this 670 sec fire but there were no sustained large-scale shifts. A set of modeling parameters was selected such that a 16,500 node Isis-3D calculation accurately reproduced the data from both experiments.

Additional experiments measured the pipe temperature under a variety of wind conditions and used different fuel pool sizes [6]. The goal of the current work is to assess how well Isis-3D, with the selected modeling parameters, reproduces the data acquired in two of those experiments. The average wind speeds in those tests were 4.6 and 2.0 m/s, but the time dependent speeds and directions exhibited more significant and sustained shifts than the 9.5 m/s test described earlier. One of these tests employed the same 18.9 m diameter fuel pool used in the earlier test, while the other used a smaller 9.45 m pool.

Fire Tests

**Facility.** A series of nine large-scale fire tests with a range of wind conditions were performed (by other investigators) at the Naval Air Warfare Center, Weapons Division (NAWCWD) fire test facility [6]. Figure 1(a) shows a plan view of the facility. An arrow shows the predominant wind direction at the site. A 3.66 m (12 ft) diameter, 18.28 m (60 ft) long culvert pipe, with 0.16 cm (0.063 inch) thick mild steel walls, was suspended approximately 0.6 m above ground level such that its axis was perpendicular to the predominant wind direction.

The facility had the ability to place two different sized pools, with diameters of 18.9 m and 9.45 m, upwind of the pipe. The downwind edges of the pools were located directly beneath the downwind edge of the pipe. At the beginning of each test the pool was partially filled with water and JP-8 aviation fuel floated on top of the water. The x-axis in Fig. 1(a) is aligned with the predominant wind direction and the z-axis is aligned with the pipe axis.

The wind speed and direction were measured at six locations upwind of the fire. The anemometers were at three elevations, 1.8 m, 5.5 m, and 9.1 m (6, 18, and 30 ft) above the ground on two poles. Figure 1(a) shows the poles were located 30 m south and southwest of the pool. The data from the 5.5 m elevation anemometers exhibited systematic time shifts relative to the other elevations. These data were not used in the current work.

The pipe interior temperature was measured at 56 locations using thermocouples. These thermocouples were equally spaced around eight different rings. In Fig. 1(a), lines on the pipe labeled 1 to 8 show the approximate axial locations of these rings. Rings 4 and 5 were 0.6 m (2 ft) apart and nearly halfway between the pipe ends. These two rings together are referred to as the central rings, and they are marked with the letter C. Left side rings (labeled 2 and 3, and together L) and right side rings (6 and 7, and together R), were place roughly 3.66 m (12 ft) on either side of the central rings. A left-end ring (labeled 1 and LE) and a right-end ring (8 and RE) were placed 6.09 m (20 ft) on either side of the central location.

Figure 1(b) shows an elevation section view of the pipe. It indicates the angular position of the thermocouples relative to the predominant wind direction. Rings 1 and 8 had only four thermocouples at positions w (windward), t (top), l (leeward), and b (bottom). Rings 2 to 7 had thermocouples at all eight positions including the wb (windward bottom), wt (windward top), lt (lee-ward top), and lb (leeward bottom) positions. At each location a 1.6 mm diameter thermocouple was pressed against the pipe interior surface using a spot welded 0.3 mm thick nichrome strip. The thermocouples were backed with 2.52 cm thick fiber insulation.

Anemometer and thermocouple data were recorded in separate computer files. One column of each file indicated the measurement time for the data in a given row relative to the first measurement in that file. However, the time scales in different files may not have been precisely synchronized.

**Experimental Wind Conditions.** This section describes wind data acquired during Experiments 3, 4, and 7 of the NAWCWD test series. Experiment 3 employed the 18.9 m fuel pool, represents a high wind speed test, and was used to select the Isis-3D modeling parameters [4]. Experiments 4 and 7 represent moderate and low wind speeds, respectively. Experiment 3 and 4 were performed using the larger, 18.9 m diameter fuel pool, while Experiment 7 employed the smaller 9.45 m diameter pool.

The thermocouple data indicated that the burn durations for Experiments 3, 4, and 7 were 670, 700, and 1100 sec, respectively. Figure 2 shows wind data measured during the burn period of each fire. The length of each line corresponds to its fire duration. These plots assume that the time scales in the wind and tempera-
ture data files for each experiment were synchronized. Figure 2(a) shows the average wind speed measured by the four anemometers on the south and southwest poles at elevations 1.8 m and 9.1 m. Figure 2(b) shows the average wind direction for the same anemometers.

The average wind speed and direction of Experiment 3 were, respectively, 9.5 m/s (22 mi/hr) and −11 deg (slightly to the left of the predominant wind direction in Fig. 1). The unaveraged data from the four anemometers (not included in Fig. 2) show considerable high frequency unsteadiness. However, the averaged data do not indicate any large-scale or sustained shifts in the wind speed or direction during Experiment 3.

The average wind speed and direction for Experiment 4 were roughly 4.6 m/s (10 mi/hr) and −10 deg. At the beginning of the test the wind blew roughly 35 deg to the left of the predominant direction (negative values of θ) with a peak speed of more than 6 m/s. It blew more nearly normal to the pipe axis from t = 300 to 600 sec with an average speed of roughly 4 m/s. After t = 600 sec the wind turned toward the left again and its speed increased.

For Experiment 7, the average wind speed and direction were respectively 2.0 m/s (4.5 mi/hr) and 40 deg to the left of the predominant wind direction. The wind blew to the right of the predominant direction for much of the first 300 seconds. It then turned increasingly toward the left during t = 300 to 600 sec, with the wind nearly stopping during t = 500 to 600 sec. The wind blew nearly parallel to the pipe axis during t = 600 to 800 sec, and it actually blew the flames away from the pipe (θ < −90 deg) during portions of that time. The wind turned to a direction more nearly normal to the pipe axis from t = 800 to the end of the test.

The wind magnitude in Experiments 4 and 7 were roughly one half and one fifth that of Experiment 3. However, the relative velocity fluctuations and direction changes in Experiments 4 and 7 were much larger than those of Experiment 3.

Computational Domain and Boundary Conditions

Figure 3 shows the Isis-3D computational domain and grid used to simulate the conditions of Experiments 3, 4, and 7. The x, y, z-coordinate axes are defined in the figure. The rectangular domain is 60 m, 15 m, and 60 m in the x, y, and z-directions, respectively. The 18.28 m long, 3.66 m diameter pipe, which was suspended 0.6 m above the bottom surface, is also shown. Different colored horizontal disks on the bottom of the domain show the locations of the 9.45 m and 18.9 m diameter fuel pools. Even though these disks are shown with finite thicknesses, the fuel vapor was injected into the domain at the bottom surface (y = 0). The pipe axis was parallel to the z-axis and the primary wind component moved in the x-direction. These directions are consistent with the coordinates used in Fig. 1(a). The computational grid uses 31, 19, and 28 volumes in the x, y and z-directions. The grid is more highly refined near the pipe than it is near the domain boundaries.

The grid on the pipe indicates the locations of surface elements. Each surface element was linked to a one-dimensional conduction module. The local heat transfer from the flowing (fire) region to each surface element was dependent on the temperature of the surface element and the computational node adjacent to it. At each time step the local heat transfer was used as a thermal boundary condition for the conduction module. The modules simulated transient conduction through the 1.6 mm thick steel pipe wall, the 1.6 mm thick thermocouple bead, a 0.1 mm air gap between the thermocouple and the nichrome strap, the 0.3 mm thick nichrome strap, and the 25.4 mm thick insulation. The surface element temperature was updated at each time step and used as a boundary condition for the flowing region of the domain. The thermocouple bead temperatures were outputs of the simulations. In this work the pipe conduction in the radial direction was modeled but the azimuthal and axial components were neglected. Moreover, even though the thermocouples and nichrome straps in the experiment were placed at discrete locations, the simulation modeled them as though they were uniformly spread over the interior surface of the pipe.

The boundary conditions employed to simulate Experiment 3 (the high wind test) were different from those used for Experiments 4 and 7. For Experiment 3, velocity boundary conditions were applied on the upwind wall (at x = 0) and on the upwind halves (x = 0 to Lx/2) of the front and back walls at z = 0 and Lz. Hydrostatic pressure conditions were applied to the remaining sidewalls. For Experiments 4 and 7, velocity boundary conditions were applied to all four sidewalls. The wind measurements acquired by both anemometers at the 1.8 m elevation were vector-
averaged and applied in the region \( y = 0 \) to 6 m. Similarly, the data measured at \( y = 9.1 \) m were averaged and applied to the region \( y = 6 \) to 15 m. Hydrostatic pressure conditions were applied to the top boundary for all simulations. No-slip velocity conditions were applied at all locations on the bottom boundary of the domain except for the fuel pool.

Fuel vapor was injected into the domain at the pool location. The large pool in Experiments 3 and 4 was divided into two regions: an interior 16.9 m diameter circle, and an outer 1 m wide ring that covered the remainder of the pool. Experiments show that the fuel evaporation rate in large hydrocarbon pool fires increases with wind speed [7]. In Experiment 3, which had an average wind speed of 9.5 m/s, the evaporation flux rates in the inner circle and the outer ring were specified to be constant at, respectively, 0.055 and 0.185 kg/m\(^2\) s, leading to an average flux of 0.084 kg/m\(^2\) s. In Experiment 4, where the average wind speed was 4.6 m/s, the inner circle and outer ring evaporation rates were 0.048 and 0.16 kg/m\(^2\) s, leading to an average rate of 0.072 kg/m\(^2\) s. The smaller pool in Experiment 7 was divided into an 8.45 m diameter inner circle with an evaporation rate of 0.024 kg/m\(^2\) s, and a 0.5 m wide ring with an evaporation rate of 0.060 kg/m\(^2\) s. These conditions lead to an average evaporation rate of 0.072 kg/m\(^2\) s. The large evaporation rate in the outer ring was intended to model high levels of heat transfer and entrainment that occur at the pool edge. However, the relative size and evaporation rates of the inner and outer regions used in this work were somewhat arbitrary.

Computational Techniques

This section summarizes the numerical and modeling methods used in the Isis-3D computational fluid dynamics/diffuse radiation computer code. A more complete description is available in Greiner and Suo-Anttila [4]. Isis-3D solves the three-dimensional mass, momentum (Navier-Stokes), energy, and species transport equations using a variable density derivative [8] of the PISO [9] pressure-based solution algorithm. It models turbulence using a Large Eddy Simulation (LES) eddy diffusivity formulation [10, 11].

Isis-3D uses a finite volume method with an orthogonal grid for discretizing the governing equations. In this formulation all vector quantities, such as heat-flux and momentum, are defined at the cell interfaces whereas scalar variables, such as temperature and pressure, are defined at cell centers. The finite volume discretization with an orthogonal grid is closely related to a finite difference approach.

Isis-3D uses a porosity method similar to the Fractional Area Volume Obstacle Representation (FAVOR\textsuperscript{TM}), a trademark of Flow Sciences Corporation, Los Alamos, New Mexico [12] method for representing curved surfaces. This method admits a flat diagonal surface within a hexahedral finite-volume computational cell. Multiple cells with diagonal interior surfaces are used to represent a curved object such as a cylinder. This representation is a much more accurate than a stair-step technique that is often used in finite difference codes. A segmented object representation has an equivalent heat transfer and flow representation as a finite-element computational-fluid-dynamics (CFD) method, but it does not incur the increased processor and memory requirements associated with finite-element methods. With the porosity method the flow areas on all cell surfaces are adjusted to account for the diagonal solid surface. Heat transfer takes place between the flowing medium and the solid in the same computational cell through the diagonal surface.

Heat Transfer Models. Within the flowing region of the computational domain, Isis-3D employs different techniques to model heat transfer inside and outside the flame zone. The flame zone is defined as computational cells where the soot volume fraction \( f_{\text{Soot}} \) is above a user-defined minimum value, \( f_{\text{Soot,min}} \). The soot volume fraction is generally larger at the center of the flame zone and decreases as the edge is approached. The minimum value therefore defines the edge of the flame zone. The effective fire volume calculated by Isis-3D increases as \( f_{\text{Soot,min}} \) decreases.

The fire interior (where \( f_{\text{Soot}}>f_{\text{Soot,min}} \)) is assumed to be optically thick and radiation transport is diffuse [13]. Diffuse radiation within the fire is modeled indirectly using the Rosseland conduction approximation. This approximation employs an effective, temperature-dependent thermal conductivity of the flowing medium equal to \( k_R = 16\pi^2\sigma T^4/3\beta_g \). In this expression, \( T \) is the local temperature, \( \sigma \) is the Stephan-Boltzman constant, \( \beta_g \) is the local extinction coefficient of the medium, and \( n \) is the index of refraction. The extinction coefficient is a function of the local mass fractions of soot, water vapor, fuel vapor and intermediate species [13]. At fire temperatures and species concentrations the Rosseland conductivity is much larger than the molecular value for air.

The optically thick assumption is not applicable at computational cells that are in contact with solid or liquid surfaces. Heat transfer to surfaces engulfed in flames is calculated based on the surface temperature and the temperature of the flow cell adjacent to that surface. Radiation resistances model the absorption within the flow cell and the surface emissivity. In Isis-3D the user inputs surface emissivities. In the current work the pipe surface emissivity is 0.9.

Outside the flames (where \( f_{\text{Soot}}<f_{\text{Soot,min}} \)), the medium does not participate in radiation heat transfer. The outer edge of the flame zone radiates heat to the environment as well as solid and liquid surfaces. Isis-3D performs a view factor calculation between all points on the fire outer surface to all points on the objects defined in the computational domain. However, shadowing from the fire itself is neglected. The heat transfer calculation assumes the flame outer surface is optically black. Heat loss to the environment is based on a user defined environment temperature. All solid objects defined in the domain also radiate to the environment when they are not engulfed in flames. The user enters the effective view factor of un-engulfed surfaces to the environment. In the current work a view factor of 0.56 is applied whenever a portion of the surface is not engulfed, and it does not vary with location or time. Future enhancements to Isis-3D will calculate this view factor at each location and at each time step.

Modeling Parameters. Isis-3D uses four separate reactions to model the chemical processes within a JP8 hydrocarbon pool fire. This model is a variant of a turbulent flame model developed by Saad et al. [14]. The mass fraction of fuel that becomes soot due to incomplete combustion and the mass fraction of fuel that becomes soot from anaerobic cracking are two parameters of the models. The pre-exponential coefficients of the Arrhenius reaction rate for the four reactions are also parameters. These six parameters were selected using soot temperature and soot volume fraction measurements acquired in a 6 m by 6 m square pool fire experiment under light wind conditions [5]. The method for determining these parameters is described in Greiner and Suo-Anttila [4]. Isis-3D simulations were then performed for a range of \( f_{\text{Soot,min}} \) and compared to data from Experiment 3 [6]. The value \( f_{\text{Soot,min}}=0.55\times10^{-6} \) (0.55 ppm) gave simulation results that most closely matched the measured data. The modeling parameters that were selected based on comparison with these two experiments were used in the validation simulations presented in the next section.

Isis-3D Validation Simulations

This section presents Isis-3D simulations of Experiments 3, 4, and 7. Simulations in this section employed no time acceleration (\( \text{TF}=1 \)) unless otherwise noted. Figure 4 contains snapshots of the fire region outer surfaces (\( f_{\text{Soot}}=f_{\text{Soot,min}}=0.55 \) ppm) from Isis-3D simulations of all three experiments. These surfaces are colored according to their local temperature. The snapshots for Experiments 3 and 4 were taken at \( t=400 \) sec (400 sec after fire
While the snapshot of Experiment 7 was taken at \( t = 700 \) sec. The green horizontal cylinders represent the pipe. The fuel pool was located in front of and to the left of the pipe, and the predominant wind direction was from the lower left of the figure toward the upper right. Movies of fire surface images show that the fire rose up from all locations of the pool at the beginning each simulation and then engulfed the pipe. After engulfment, the surfaces exhibit large scale puffing. However, the surfaces in Fig. 4 are fairly typical of other times during the simulations.

The strong winds in Experiment 3 blew the majority of the simulated fire volume downwind of the pipe. The fire was wide enough to almost continuously engulf the entire length of the pipe. However, the upper portion of the windward surface was intermittently outside the flames. The lighter winds in Experiment 4 allow the fire volume to rise upward above the pool. Movies show that the central region of the pipe was almost continuously engulfed in flames but that the ends were frequently outside the fire zone. The light winds and small pool size of Experiment 7 allowed the majority of the fire volume to stay near the center of the windward side of the pipe. The wind direction changed significantly during this simulation, but the pipes ends were never engulfed in the fire volume.

For all three simulations the fire volumes were not fully contained in the computational domain and they exited through the upper and right side boundaries of Fig. 3. However, for the portions within the domain, the average fire surface temperatures for Experiments 3, 4, and 7 were roughly 960 K, 920 K, and 1030 K, respectively. These temperatures are consistent with experimental measurements that indicate that the emissive power weighted averaged surface temperature of hydrocarbon fuel fires with diameters of 20 m and 10 m are approximately 940 K and 1070 K, respectively [15]. The current simulations of Experiments 3 and 4 also suggest that the surface temperature may increase when strong winds are present.

**Experiment 3.** Figure 5 shows local thermocouple temperatures for the central regions of the pipe (rings 2 to 7) versus time from Experiment 3 (data for the left end LE and right end RE rings are not included). Figure 5(a) shows measured data while Fig. 5(b) presents results from the Isis-3D validation simulation. The simulations used a computational grid that was not sufficiently refined to resolve the difference between ring pairs 2 and 3 (the left rings, L), 4 and 5 (the central rings, C), and 6 and 7 (the right ring, R). In Fig. 5(a), the data with the prefix \( C \) are averages of the measurements from rings 4 and 5, and the suffixes indicate the angular location as seen in Fig. 1(b). This averaging is used so that the experimental data can be easily compared to the simulation results. Similarly, the prefixes \( L \) and \( R \) indicate data averaged from rings 2 and 3, and rings 6 and 7, respectively.

Figure 5(a) show that all the measured temperatures were roughly 300 K before the test began. The temperatures at different locations did not begin to rise at the same time. Presumably this was because the fire was started at discrete locations in the fuel pool and required time before it spread to cover the pool. The temperature of all the thermocouples rose until roughly \( t = 300 \) sec when they reached steady state conditions. The steady-state temperatures ranged between 1000 K and 1600 K. Finally, the temperatures on the central C and L left rings began to increase again at time \( t = 450 \) sec, and they became nearly uniform at 1500 K after \( t = 600 \) sec. Surprisingly, the wind data in Fig. 2 do not exhibit any sustained or large-scale shifts during that time period.

Figure 5(b) shows local thermocouple temperatures versus time from the Isis-3D validation simulation for Experiment 3. The initial pipe temperature was 300 K. The pipe temperature began to...
rise at $t=0$ for all location because the simulated fire zone rose uniformly from all locations of the fuel pool. At all times after $t=0$ the range of temperatures in the simulation was similar to that of the experiment. However, the simulated temperatures at individual locations were not the same as the experiment. The simulations do not exhibit the secondary temperature rise beginning at $t=450$ sec seen in the experiment. This is not surprising since the wind conditions did not show evidence of a sustained change at that time. The secondary rise is not currently understood. It may have been caused by boiling of the water beneath the fuel layer at that time. The secondary rise is not currently understood. It may have been caused by boiling of the water beneath the fuel layer at the end of the fire, or some other condition that was not modeled in the simulation.

Figure 6 shows the angular variation of temperature at time $t=400$ sec for Experiment 3. Figure 5 shows that the temperatures are relatively steady at that time. Different colored lines indicate data from different axial positions. Measured data are shown in Fig. 6(a), while Isis-3D results are presented in Fig. 6(b). In both the experiment and the simulation the lowest temperatures are near the top of the pipe and the highest temperatures are on the leeward side. While the highest measured temperatures are located on the upper portion of the leeward side, the highest temperatures from the simulations are nearer to the ground. The strong winds that were present during this test tilted the flames so that the leeward surface of the pipe was more continuously engulfed in flames than the top. Moreover, a recirculation zone may have developed downwind of the pipe. Enhanced air/fuel mixing in such a zone may increase the fire temperatures, which would contribute to the high temperatures observed on the leeward pipe surface. Finally, the temperatures in both the experiment and the simulation are fairly uniform at each axial position. This is consistent with the axial flame coverage observed in Fig. 4(a).

**Experiment 4.** Figures 7(a) and 7(b) show the measured and simulated thermocouple data versus time for Experiment 4. Figure 7(a) indicates that the temperature rise rate of several of the right side thermocouples (black lines) was significantly slower during the first 250 sec of the fire than it is after that time. This is particularly true for the top and leeward regions of the pipe. Moreover, the temperature of the right side thermocouples that were not near the bottom of the pipe decreased after $t=550$ sec. The direction data in Fig. 2(b) shows that the wind blew to the left before $t=250$ sec. It then shifted to a direction more nearly normal to the pipe axis. At $t=600$ sec the wind turned to the left once again and the wind speed increased. These shifts in the wind speed and direction are nearly synchronized with the changes in the right side temperature rise rates. Finally, the measured steady state temperatures (after $t=400$ sec) ranged between 1000 K and 1550 K.

The simulation results presented in Fig. 7(b) show a number of similarities with the measured data. The temperature rise rates for the top and leeside thermocouples on the right side of the calorimeter (black lines) were significantly greater after $t=250$ sec than they were before that time. Most of the right side temperatures also decreased after $t=550$ sec. All but one of the simulated steady-state temperatures (after $t=400$ sec) were between 1100 and 1650 K.

Figure 8 shows the radial variations of the measured and simulated temperatures at $t=400$ sec. The time dependent wind and temperature conditions in Figs. 2 and 7 were fairly constant at that time. The experimental data in Fig. 8(a) show that the temperatures on the left end, left and central (LE, L and C) rings were significantly higher than they were on the right and right end (R and RE) rings. Moreover, the temperatures on the windward side were slightly higher than they were on the leeside. In the validation simulation, the left and central (L and C) rings are the hottest, and the left end ring (LE) was the coolest. These results are consistent with the measured data. However, the simulated left end
temperatures are cooler than the measured data. Even though the wind blew fairly strongly to the left for much of the time preceding \( t = 400 \text{ sec} \), the simulated fire did not appear to extend as far to the left as the actual fire. Finally, the simulated temperatures at \( t = 400 \text{ sec} \) were slightly hotter on the leeside of the pipe than they were on the windward side.

**Experiment 7.** Figure 9 shows measured and simulated thermocouple data versus time for Experiment 7. Compared to the other tests, Experiment 7 employed the smallest fuel pool, it was conducted under the lightest wind conditions, and it had the longest fire duration. The relative variations of the wind speed and direction were also the largest of the three experiments.

The experimental data in Fig. 9(a) show that the temperatures generally rose for the first 350 sec of the test. At \( t = 370 \text{ sec} \) some of the left side temperatures (pink lines) began to rise rapidly and some of the right side temperatures (black) decreased. The direction data in Fig. 2(b) show that the wind turned from the right to the left at that time. The wind blew parallel to and away from the pipe axis during the period \( t = 600 \) to 800 sec. All but one of the thermocouples temperatures that were not on the bottom of the pipe decreased during that time. The multiple periods of increasing and decreasing temperatures in Fig. 9(a) appear to be the results of the rapidly changing wind speed and direction during the test. The measured steady state temperatures were generally between 600 and 1350 K after \( t = 600 \text{ sec} \).

The simulated thermocouple temperatures in Fig. 9(b) exhibit a number of periods of increasing and decreasing temperatures, similar to that of the measured data. The simulated thermocouple temperatures increase for the first 340 sec of the fire before dropping off. However, the simulated temperatures at individual locations were not the same as the measured values. In particular, the simulation displayed temperature increases starting at \( t = 470 \text{ sec} \), particularly on the bottom of the right side, that were not exhibited by the measured data. The simulated temperatures after \( t = 400 \text{ seconds} \) ranged between 500 and 1750 K, which is somewhat higher than the measured range.

Figure 10 shows the radial variations of the temperatures averaged over the period \( t = 400 \) to 600 sec. The measured data in Fig. 10(a) show significant differences at each axial ring. The left and central (L and C) rings were the hottest during that period and the right end (RE) ring was the coolest. Moreover, the windward portions of the left, central and right rings were hotter than their leeward sides. The simulation results also show that the left and central rings were the hottest and the right end ring was the coolest. However, the simulation indicates that the highest temperatures were on the leeside of the pipe. A similar comparison of the temperature distributions for the time period \( t = 600 \) to 800 sec shows significant differences between the measured and simulated results.

Experiment 7 was the most difficult of the three to simulate. The varying wind conditions contributed to this. Extensive portions of the pipe were outside the flame zone during most of this experiment. This made view factor radiation more important for Experiment 7 than it was for the others.

**Average Temperatures.** Figure 11 shows the average thermocouple temperature rise versus time for all three experiments. This rise is defined as the average temperature of all 56 thermocouples at time \( t \) minus the value at \( t = 0 \). This temperature rise is approximately proportional to the total heat delivered to the pipe as a function of time. Thermal analysis tools used in risk studies must accurately predict the total heat delivered to hazardous material packages from a fire. This information is used to predict if internal components reach their limit conditions \( [16,17] \).

The thickest lines in Fig. 11 show experimental data for Experiments 3, 4, and 7. Thinner lines show results from Isis-3D simulations.
for each experiment. Results from simulation that employed acceleration time factors of TF=1, 4 and 12 are included for Experiment 3, while TF=1 and 4 results are presented for Experiments 4 and 7.

The experimental data show that the average pipe temperature increased rapidly for the first 300 sec of each test before approaching steady state. Experiments 3 and 4 employed the larger 18.9 m diameter pool. Their steady state average temperatures were significantly higher than that of Experiment 7, which employed the smaller 9.45 m diameter pool. Experiment 3, which experienced higher wind speeds, exhibited higher average temperatures. The recirculation zone downstream from the pipe, which intensifies the mixing of fuel vapor with air and increases the fire temperature, may have caused the higher pipe temperature.

For Experiments 3 and 4 the TF=1 simulation results are in good agreement with the experimental data for all times. For Experiment 7 the TF=1 simulation diverges from the experimental data during the period 525 sec < t < 1000 sec. Figure 2 shows that the wind blew the fire only mildly toward the pipe during this period so that much of the pipe was not engulfed in flames. Figure 9(a) shows that most of the measured temperatures decreased during this period but some of simulated temperatures in Fig. 9(b) increased. However, viewed as a whole, Fig. 11 shows Isis-3D accurately predicts the total heat transfer from fires to the pipe under a fairly broad range of wind conditions.

The average temperature rise from accelerated simulations is also included in Fig. 11. These calculations produced average temperature data that were in good agreement with the TF=1 simulations. The average temperatures from the accelerated simulations generally exhibited larger amplitude periodic oscillations than the TF=1 calculation. As described earlier, these simulations employed pipe and thermocouple specific heats that were greatly reduced below their physical values. This caused the pipe temperatures to over respond to the unsteady fire puffing, and lead to the oscillations.

For Experiment 3, the TF=1 simulation required 40 hours on a 2.4 GHz personal computer with 0.5 Gb of RAM. The TF=4 and 12 calculations required only 10 and 3.5 hours on the same machine. Simulation results with TF=4 are also in good agreement with TF=1 data for Experiments 4 and 7. However, TF=12 results (not included in Fig. 11 to avoid overcrowding) deviated substantially from the TF=1 results. The wind conditions in Experiments 4 and 7 were significantly more unsteady than those in Experiment 3. It appears that TF=12 simulations are only accurate if the wind conditions are reasonably steady.

Conclusions

The Isis-3D computational fluid dynamics/radiation heat transfer code has been developed to perform engineering level simulations of heat transfer from fires to hazardous material packages. It was designed to provide reasonably accurate estimates of the total heat transfer from large fires to engulfed or nearby objects under a variety of circumstances, reproduce the general characteristics of the object temperature, and require fairly short computer turnaround times.

In this paper validation calculations were performed to assess the ability of Isis-3D to reproduce measured data from three large-fire experiments. Those experiments measured the temperature of a large culvert pipe suspended over the leeward edge of hydrocarbon pool fires under a variety of crosswinds. The tests used two different fuel pool diameters (18.9 m and 9.45 m) and three different average wind speeds (2.0, 4.6 and 9.5 m/s). The lowest wind speed test exhibited the highest levels of wind unsteadiness among the three experiments, and the highest speed test displayed the least unsteadiness. Isis-3D accurately reproduced characteristics of the time-dependent local and average temperatures for all three tests. It more faithfully reproduced the data from the high wind speed, low unsteadiness tests compared to the low speed, highly unsteady test.

Accelerated simulations were performed in which the pipe specific heat was reduced compared to the physical value by a factor of four. This artificially increased the speed at which the pipe temperature rose and allowed the simulated fire duration to be reduced by a factor of four. A 700 sec fire with moderately unsteady wind conditions was accurately simulated in 10 hours on a 2.4 GHz LINUX workstation with 0.5 GB of RAM.
Acknowledgment

This work was supported by the Defense Threat Reduction Agency (DTRA) through a Small Business Innovative Research project, contract number DTRA01-01C-0084 (Phase II SBIR).

Nomenclature

- $f_{\text{Soot, min}}$: soot volume fraction at flame boundary
- $f_{\text{soot}}$: soot volume fraction
- $L_X L_Y L_Z$: domain dimensions
- $N_X N_Y N_Z$: number of cells in x, y and z directions
- $s$: wind speed
- $T$: temperature
- $t$: time
- $x, y, z$: Cartesian coordinates
- $\Delta T_{\text{avg}}$: average thermocouple temperature rise
- $\theta$: wind direction

References


