Simulation Method for the Fire Suppression Process Inside the Engine Core and APU Compartments

Jaesoo Lee, Ph.D.
The Boeing Company
P.O. Box 3707, MC 67-ML, Seattle, WA 98124-2207

ABSTRACT

Fire dangers exist inside engine core and auxiliary power unit compartments of commercial airplanes where jet fuel or lubricant oil leaks can catch fire on hot engine surfaces. Fire suppressants need to be quickly dumped into the compartments to prevent any eruption of fire. To quickly deliver fire suppressant from a storage bottle through a distribution pipe and injection nozzles to vented engine compartments, a fire extinguishing system should be designed and installed effectively. In certification tests of a fire extinguishing system, the optimum installation of multiple injection nozzles is an important factor contributing to successful fire suppression. At the present time, the selection of the nozzle locations and orientations for fire extinguishing systems depends primarily on a series of ground tests continued until the optimum installation conditions are found that satisfy the minimum performance standard. The certification test process is time consuming and relies greatly on empiricism.

This paper deals with a simulation method for a fire suppression process that is based on the computational fluid dynamics method of modeling the two-phase flow physics. The method has been applied to simulate the tests of engines, auxiliary power units, and the FAA's Nacelle Fire Simulator. Analysis results so far reveal excellent correlation with the test data, demonstrating the validity of the simulation method. It is anticipated that the method would serve as a useful tool not only for designing efficient firex systems, but also for guiding certification tests.

INTRODUCTION

The current designs of fire extinguishing (firex) systems for engine nacelles and auxiliary power units (APU) of commercial and military airplanes and helicopters depend primarily on experience, and the optimum installation of the agent injection nozzles requires a series of ground tests to verify the capability of putting out a fire inside compartments and to meet the FAA's certification requirements. The ground tests cover tests at different bottle temperature conditions to ensure the satisfactory operation of the firex system and the required agent concentration level at any weather and airplane operation conditions. The testing is usually time-consuming, and systems can easily be over-designed, resulting in an excess of the firex agent. To design efficient firex systems and to save the time for the certification tests for new engines and APUs, a scientific design tool is necessary that has the capability to simulate the entire system and the fire-suppression process using Halon 1301 and Halon replacements.

This paper describes the simulation method, example simulations conducted based on available firex performance test data, and some conclusions of the work so far. Important simulation factors were identified that affect the prediction accuracy; they are also briefly described.

Engine Fire / Overheat Detection and Fire Extinguishing

Figure 1 is a diagram of the processes of engine fire-and-overheat detection and fire extinguishing. Each engine has fire detectors that monitor the engine for fire and overheat conditions and supply temperature data. These detectors send signals for flight deck indicators including lights in the fire switch, the fuel-control switch, fire warning aural, and master warning lights. Two fire extinguishing bottles with Halon 1301 are located in the forward cargo compartment. Pipes connect both bottles to discharge nozzles in each engine compartment. When a fire switch is pulled by a crewmember because of a fire detected in an engine, fuel supply to the engine stops; when the fire switch is turned, Halon 1301 discharges from one of the bottles and flows to the engine.

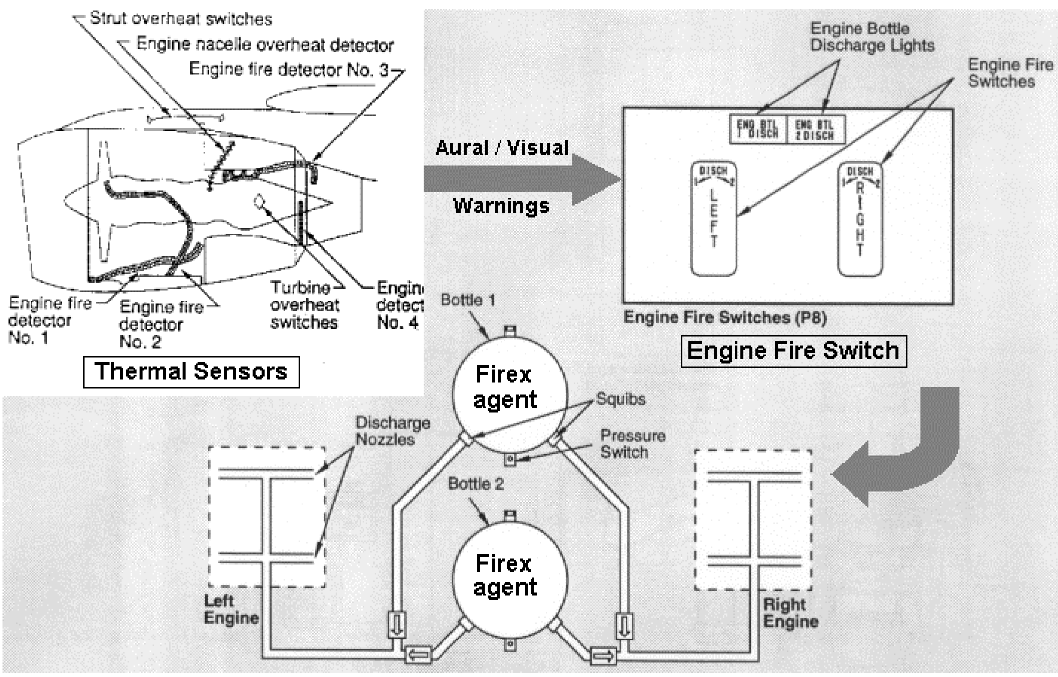
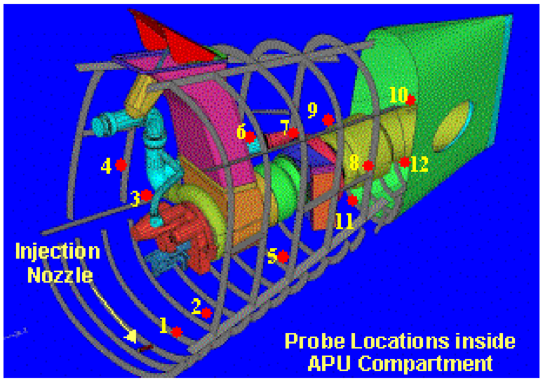


Figure 1: Engine Fire-and-Overheat Detection and Fire Extinguishing

The fire-detection and warning process for APUs is similar to that for engines. However, when an APU's fire-detection system warning is signaled, the APU shuts down automatically. The APU fire-extinguishing bottle, which contains Halon 1301, is on the forward side of the APU compartment firewall. When the airplane is on the ground and the engines are off, an APU fire causes automatic bottle discharge. In flight, when the crew pulls the APU fire switch in the flight deck, the APU fuel- and air-shutoff valves close. When the crew turns the APU fire switch, the bottle discharges Halon 1301 into the APU compartment.

Certification Requirements for Firex System

The current FAA Advisory Circular 20-100 [Ref. 1] specifies the requirements for the firex systems of engines and APUs using Halon 1301. The requirement is that the concentration level should be greater than 6% by volume throughout the protected zone within compartments for longer than a half second simultaneously. Figure 2 shows 12 concentration probe locations inside an APU compartment. The green rectangle in the plot at the right shows a successful certification test. When Halon replacements or alternatives are in use, the height of the acceptance window will change due to different characteristics of physical properties and chemical reaction with fire. The FAA is testing different suppressant agents using the Nacelle Fire Simulator (NFS) to experimentally determine the Minimum Performance Standard (MPS) for fire suppression.



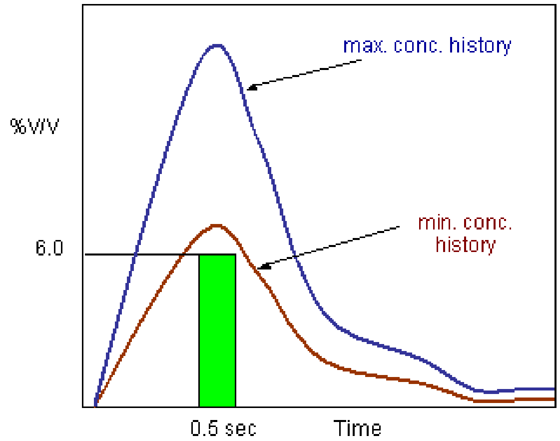


Figure 2: Concentration Probe Locations and Range of Concentration Histories

Environmental and Physical Properties of Agents

Halon 1301 ($CBrF_3$) is an efficient fire suppressant that has been widely used in firex systems for engines and APUs. However, production of Halon has been banned in the United States since 1994 because of negative impacts to the environment—ozone depletion, global warming, and long atmospheric life. Comparison of some environmental and physical properties of Halon 1301 and those of other fire suppressants (HFC-125, CF_3I) is shown in table 1. No selection of Halon replacement has been made yet for engines and APUs of commercial airplanes, but the selection of a new fire suppressant will impact the design and the certification of a new firex system.

Table 1: Environmental and Physical Property of Fire Suppressant Agents

Properties	Halon 1301	HFC-125	CF_3I
Chemical formula	$CBrF_3$	CF_3CHF_2	CF_3I
Ozone depletion potential	16	0	0.0002
Molecular weight	148.9	120.0	196.9
Global warming potential	5600	2800	5
Critical temperature, °F	152.6	151.3	251
Atmospheric lifetime, years	65	33	0.0137
Liquid density, lbm/ft^3	96.01	74.3	131.4
Boiling point, °F	-72	-55	-9
Heat of vaporization, Btu/lbm	35.5	70.7	48.1
Vapor pressure at 77 °F, psia	234.8	200.4	63.7

DESCRIPTION OF THE SIMULATION METHOD

Figure 3 is a schematic picture of a firex system and a vented compartment considered in the present simulation method. The firex agent is stored in a storage bottle, which is usually pressurized by nitrogen in the range of 500 to 900 psia. Therefore, when the liquid agent and nitrogen mixture is discharged from the bottle, the relative contents of the liquid- and the gas-phases of the agent, the pressure, and the temperature at the exits of injection nozzles varies with time. The two-phase agent flow mixes with various sources of vented air inside the engine core and APU compartments in the high-temperature but non-pressurized environment.

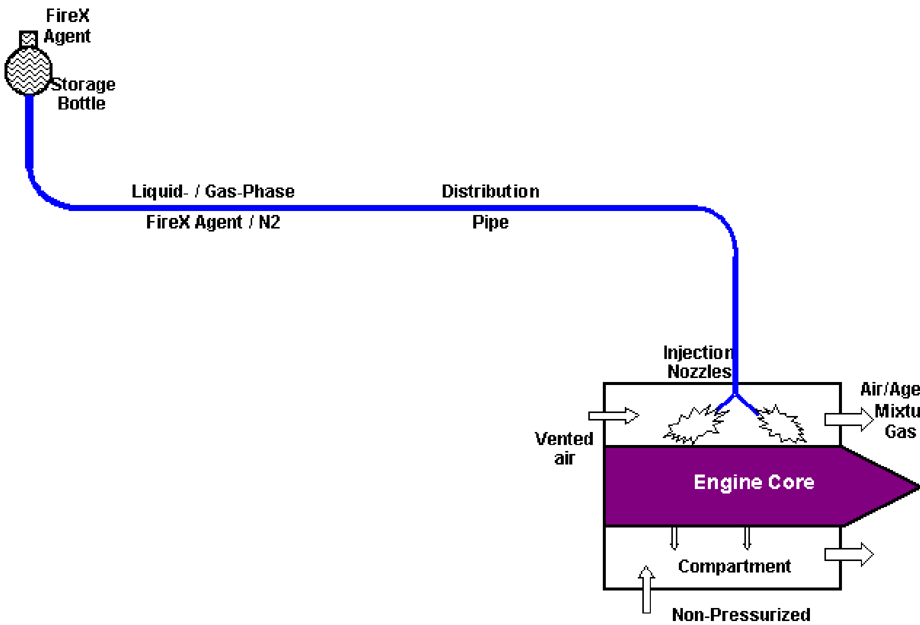


Figure 3: Schematic of Simulated Fire Extinguishing Process

A block diagram for the analysis of the discharge of agent from a storage bottle through injection nozzles and the prediction of concentration histories inside a vented compartment is shown in figure 4. The effective simulation of the process is a challenging task, due to the complex geometry of the engine core and APU compartments, the uncertainties in many airflow sources inside the compartment, the complicated two-phase flow physics of the agent jet from injection nozzles, and the related difficulties in the CFD analysis.

Firex System Analysis

The agent flow conditions at injection nozzles during the injection period are analyzed first using a one-dimensional, two-phase flow code, Hflowx, which simulates the discharge of agents (Halon 1301, HFC-125, or CF_3I) and nitrogen from a storage bottle through pipes and nozzles. Hflowx is an extension of the Hflow code [Ref. 2], which was developed for applications using only Halon 1301. The predicted unsteady agent flow conditions are used as the transient boundary conditions in the subsequent CFD analysis to predict concentration distribution inside a compartment.

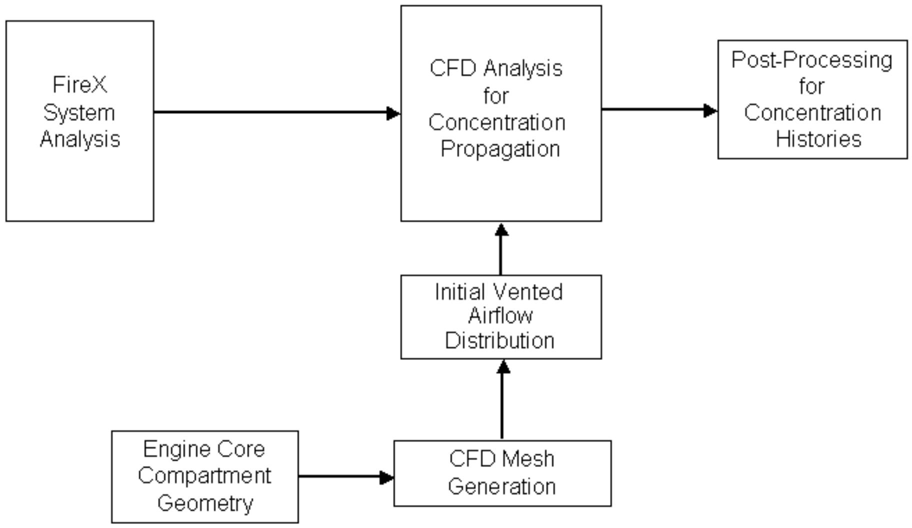


Figure 4: Block Diagram of Elements of the Developed Simulation Process

Figure 5 shows the Hflowx analysis of the discharge of Halon 1301 and nitrogen from a storage bottle through pipes and nozzle. Input parameters include the suppressant agent name, the storage bottle volume, the charging pressure and the temperature of the storage bottle, the test temperature, the agent mass, and the piping information. The code employs an iterative time-marching scheme to solve the quasi-one-dimensional conservation equations for mixture flows from the storage bottle to the pipe end. The constitutive equations include the thermo-physical property relationships for the liquid, the vapor, and the mixture fluid, and the equation of state for an ideal gas. The predicted mass flow rates of (1) the vapor and liquid phases of the agent and (2) the mixture pressure and temperature at the exit of the injector are used as the time-dependent boundary conditions at the interfaces between the nozzles and the vented compartment.

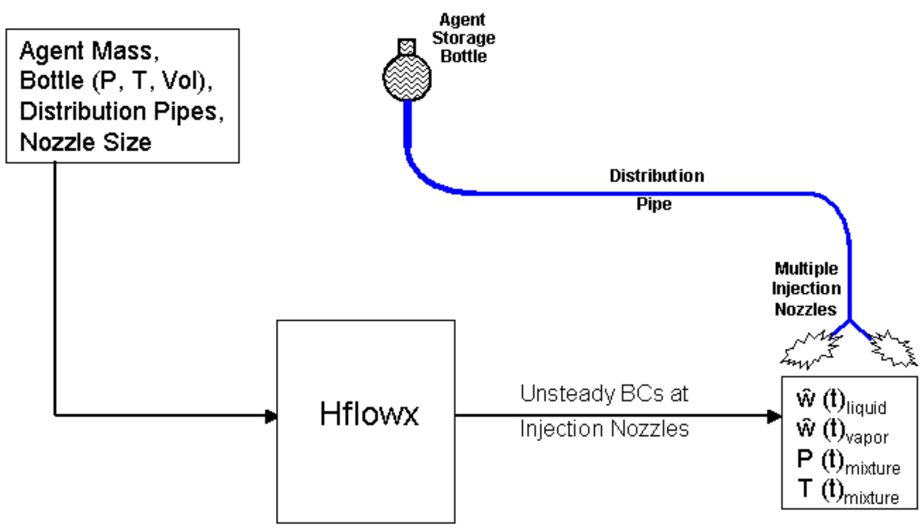


Figure 5: Hflowx Predictions of Flow Properties at the Injection Nozzle Exit

The capability of Hflowx was evaluated using the bottle pressure drop test data obtained from the firex system of the FAA Nacelle Fire Simulator. The test flow conditions and the geometrical data in table 3 were used to predict the unsteady bottle pressure decay. The comparison of the predicted and the measured bottle pressure decay was in excellent agreement (figure 6).

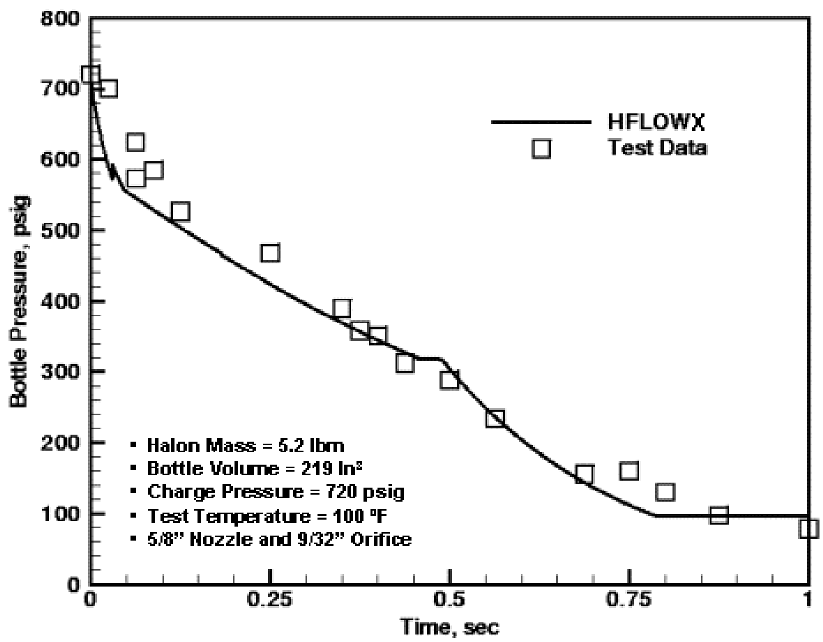


Figure 6: Comparison of Predicted and Measured Bottle Pressure Decay

Figure 7 shows the predicted discharge characteristics of Halon 1301, HFC-125, and CF_3I using the design and test conditions of a selected firex system; the figure reveals that discharge takes an increasingly longer time with HFC-125, Halon 1301, and CF_3I, in that order. Also shown is that CF_3I has a higher content of liquid-phase agent coming from the injector nozzle compared with the other agents tested, which was anticipated because of the higher boiling point for CF_3I as shown in table 1.

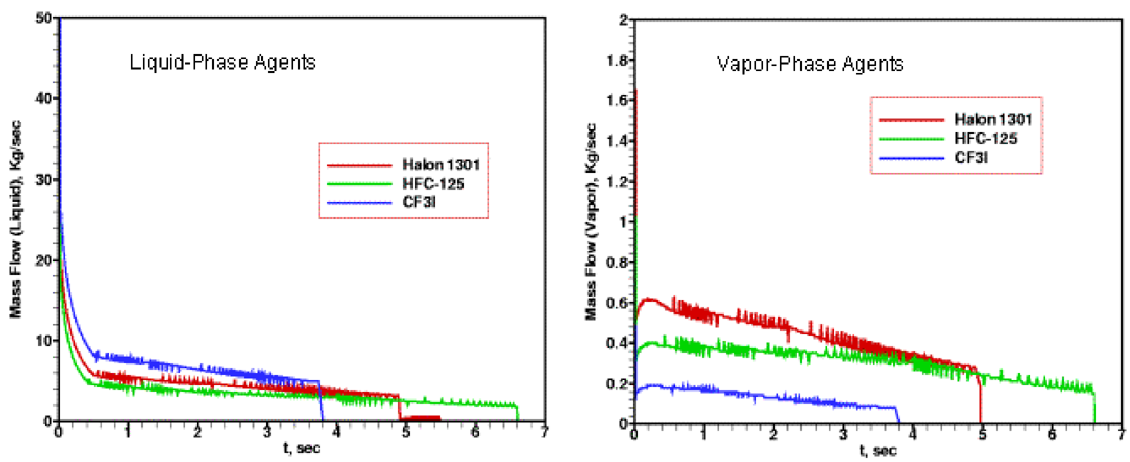


Figure 7: Comparisons of Discharge Characteristics of Firex Agents at Nozzle Exit

Mesh Generation

The computational meshes for CFD analyses are three-dimensional unstructured meshes, which are adequate to model the complex geometry of engine and APU compartments. The solution domains should be selected based on the flow physics phenomena during the entire fire suppression process. Therefore, the boundaries of solution domain should be defined considering not only the vented airflow but also the propagation of agent concentration during the entire fire suppression process. The *icemcfd* code [Ref. 3] was used to generate the three-dimensional unstructured mesh. The injection nozzles were also included in the mesh generation to model the injection of agent. Figure 8 shows the typical unstructured surface mesh inside the engine core and APU compartments. Thin wires, tubes, and small parts on the engine surfaces, which are considered to be fluid dynamically insignificant, were removed from the digitized geometry database before mesh generation. Less than a half million tetrahedral cells were required for the engine core or the APU compartment. The grid adaptation scheme was also employed to refine the meshes in the regions of higher gradient during the solution process.

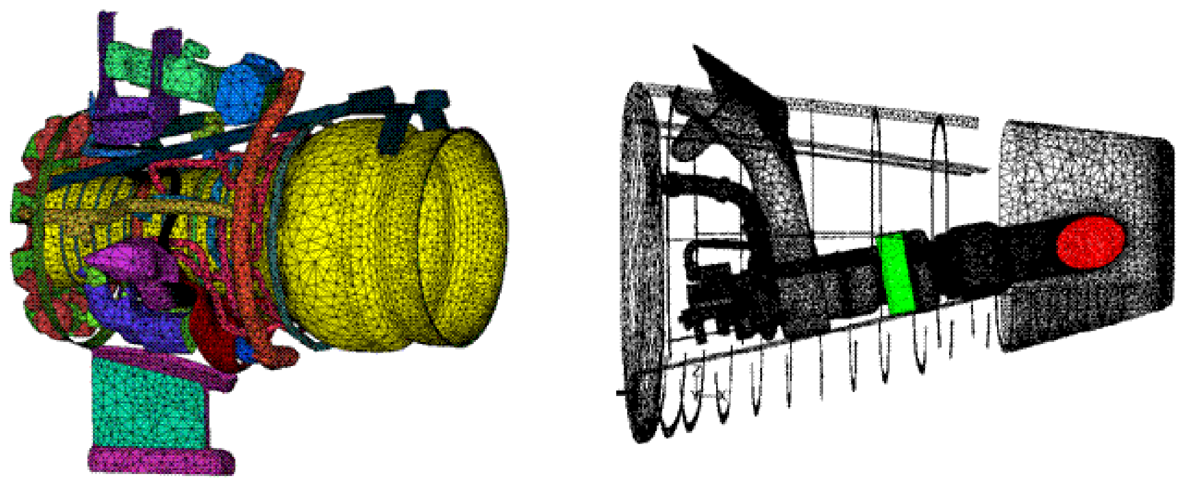


Figure 8: Unstructured Surface Meshes Inside the Engine Core and APU Compartments

Initial Vented Airflow Distribution

The unsteady analysis of concentration distribution requires the initial airflow distribution inside a compartment just before the agent injection. This airflow distribution can be obtained using the CFD approach by solving the mass continuity equation, the momentum equations, the thermal energy equation, the turbulence model equations, the radiative heat transfer model equation, and the equation of state. The buoyancy effect was also accounted for in the analysis. Some of the important boundary conditions are the airflow conditions at a variety of vented air sources, the temperatures and heat transfer properties of the solid surfaces inside a compartment, and a non-slip viscous wall boundary condition. Some flow obstacles (e.g., a heat exchanger unit, flanges) need to be modeled to account for the changes in the flow properties along the airflow path and the flow residence time.

CFD Analysis for Concentration Propagation

The CFD analysis is based on the unsteady solution of conservation equations of the mass, the momentum, and the energy transports of vent air and firex agent mixture inside the complicated three-dimensional compartment geometry. In addition, the species transport equation and the turbulent model equations are solved to calculate the agent concentration (mass fraction) distribution in the air and agent-vapor mixture flow and the turbulent flow quantities required in the transport equations. The Spalart-Allmaras one-equation turbulence model [Ref. 4] was used in the present simulations to save analysis time. The Fluent [Ref. 5] Navier-Stokes code was used in the CFD simulation of the process inside the compartments.

The vapor-phase and liquid-phase agent mixture from injection nozzles was modeled using the combined Eulerian and Lagrangian modeling schemes [Ref. 6]. Figure 9 shows the details of the governing equations in the unsteady CFD simulation of agent injection and concentration propagation inside a compartment. In the present simulation method the liquid-phase agent flow from an injection nozzle was treated as a stream of spherical droplets with a size distribution. Therefore, the droplets were modeled using the Lagrangian conservation equations. Three assumptions were made: First, the vapor-phase agent fluid was treated as an ideal gas; second, the volumetric effect of liquid droplets in the mixture flow inside the compartment was assumed negligible; and third, droplets rebound from the solid surface.

The Eulerian transport equations of the gas mixture and the Lagrangian equations of the discrete-droplet phase were solved fully coupled to account for the mutual interaction between the evaporating and moving suppressant droplets and the gaseous suppressant/air mixture.

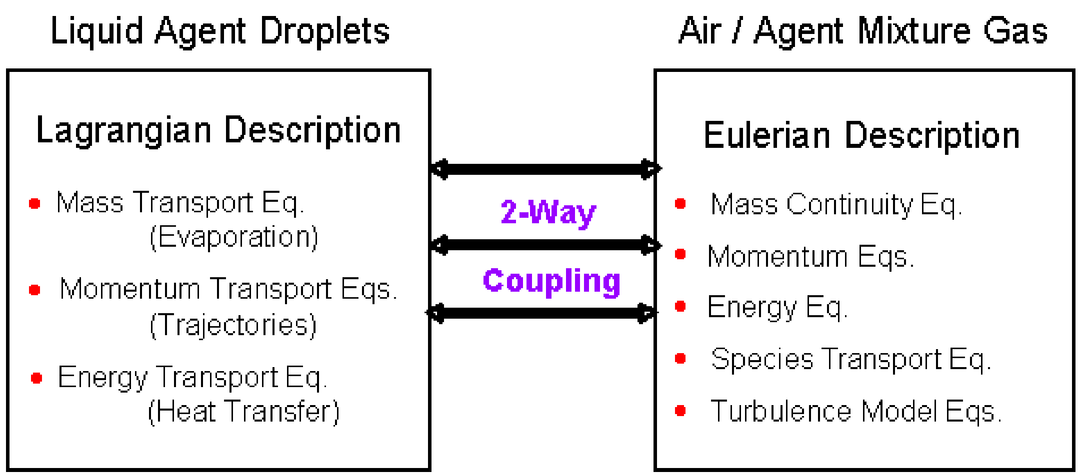


Figure 9: Types of Governing Equations in the CFD Modeling of the Agent Injection and Concentration Propagation Processes

The thermodynamic and transport properties of agent vapor and liquid droplets are required as the input to the CFD code in addition to the similar properties for air. The properties are, in general, a function of temperature. For a gaseous agent, the thermal conductivity, the specific heat, the dynamic viscosity, the molecular weight, and the Lennard-Jones characteristic length and energy parameter were required. For a liquid droplet, the required properties are density, thermal conductivity, specific heat, viscosity, latent heat, vaporization temperature, boiling point, binary mass diffusivity, saturation vapor pressure, and surface tension. For the thermodynamic properties of the air and agent gas mixture, ideal-gas mixing laws were employed.

The CFD analysis of the fire-suppression process performed with more stability using the double-precision computation and the 2nd-order temporal and spatial differencing schemes. The time steps for the stable unsteady analysis should be varied with the stages of the fire suppression process. In general, a time step on the order of 10^{-4} sec was required during the initial stage of agent injection. After the injection was over, the time step could be increased from 0.001 sec to 0.01 sec.

Post-Processing for Concentration Histories

During the transient CFD analysis of the fire suppression process, the predicted mass fractions of the agent at 12 concentration probe locations were continuously sampled at the end of each computational time step. The concentration in mass fraction was then converted to volumetric fraction using the following relation:

$$\alpha_v = f / [f + (1 - f) (M_h / M_a)]$$

where, α_v is volumetric fraction and f, M_h, and M_a denote the mass fraction of the agent gas, the molecular weight of the agent gas, and the molecular weight of air, respectively.

The variations of volumetric fractions with time at 12 probe locations were continuously recorded in output files, which were used to make a concentration-histories plot at the completion of an analysis. The concentration history plot is used to determine whether the predicted concentration histories meet the FAA's certification requirements.

EXAMPLE SIMULATIONS

Two example simulation cases are described in this paper that demonstrate the capabilities of the developed simulation methods. The first case was the simulation based on the firex test conducted using the FAA Nacelle Fire Simulator; the second case was for the APU compartment of the Boeing 777 airplane. The predicted results are discussed and compared with the experimental test data.

FAA Nacelle Fire Simulator Case

A schematic picture of the test section of the NFS is shown in figure 10 including important parts inside the vented compartment. The diameter of the test section is approximately 4 ft. Vented airflow enters the Nacelle Fire Simulator from the left side of the test section. The fire suppressant agent is injected against the air stream using two pairs of nozzles and orifices on the two sides of the test section, as shown in figure 10.

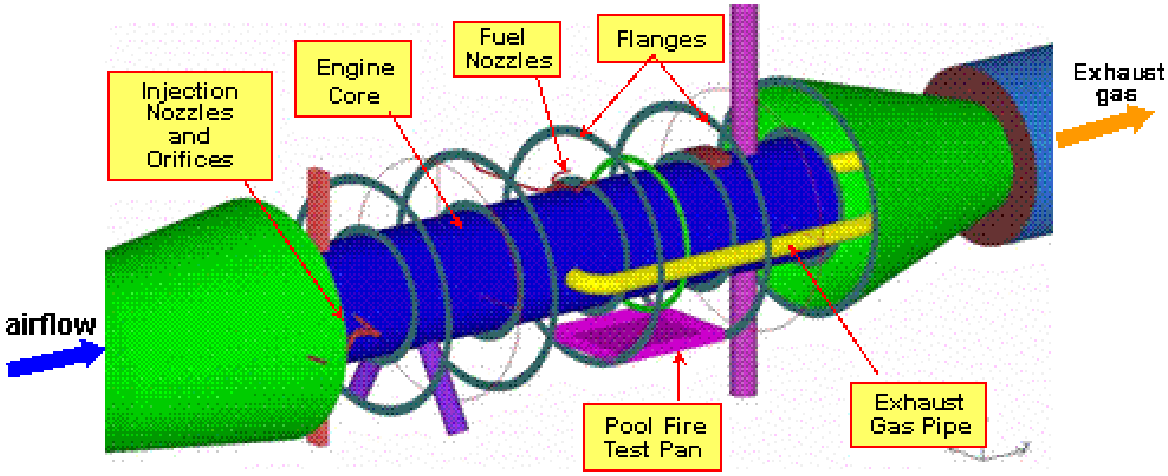


Figure 10: Picture of the Internal Test Section of the FAA Nacelle Fire Simulator

The incoming airflow mixes with the injected agent jet inside the compartment and the gas mixture leaves the exit at the end of the test section. The airflow analysis during steady-state engine operation was conducted at the flow conditions in table 2. The predicted flow and temperature distributions were used as the initial conditions in the unsteady analysis of concentration distribution of Halon 1301 inside the compartment.

An unstructured surface mesh used for the CFD analysis of the NFS flow field is shown in figure 11. The airflow inlet boundary was defined at approximately 12.5 ft upstream of the injection nozzles. The number of computational cells was approximately 350,000.

Table 2: Firex Test Conditions of the NFS

Items	Conditions
Total vented airflow	2.2 lbm/sec
Total injected agent mass	5.2 lbm
Storage bottle volume	219 in^3
Bottle charge	812 psia, 70°F
Discharge test conditions	100 °F, 14.7 psia
Injector inner diameters	0.525” (2 nozzles) 0.281” (2 orifices)

The Hflowx code was used to predict the Halon 1301 flow conditions at the exits of four injectors using the firex system design and the operating conditions in table 2. The predicted agent flow conditions at the injection points - the mixture pressure, the mixture temperature, the liquid-phase mass flow rate, and the vapor-phase mass flux - are shown in figure 12.

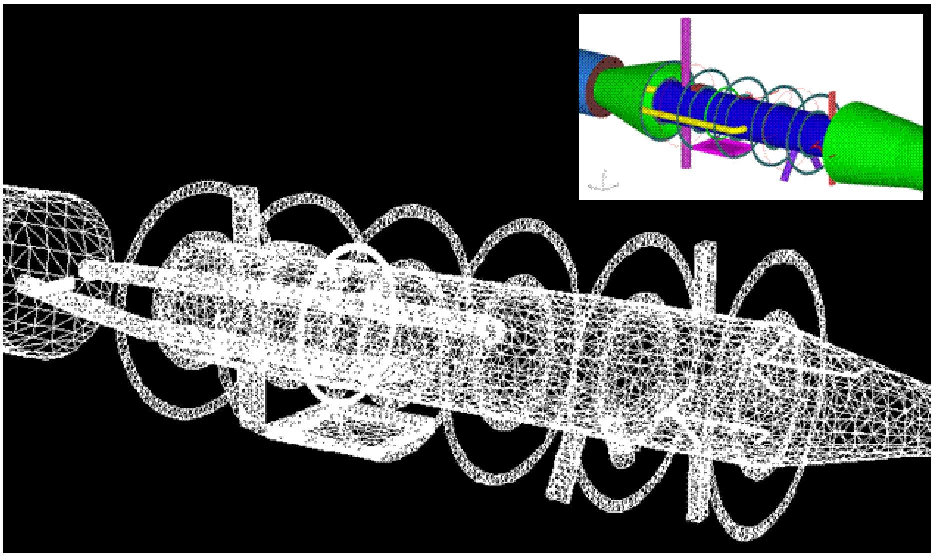


Figure 11: Unstructured Surface Mesh for the Nacelle Fire Simulator

Figure 13 shows a series of concentration distributions during the initial 45-millisecond period at an axial station located 4 inches upstream of the four injection nozzle exits. Figure 13 clearly illustrates how the four discrete jets grow and mix with the counter-flowing air stream.

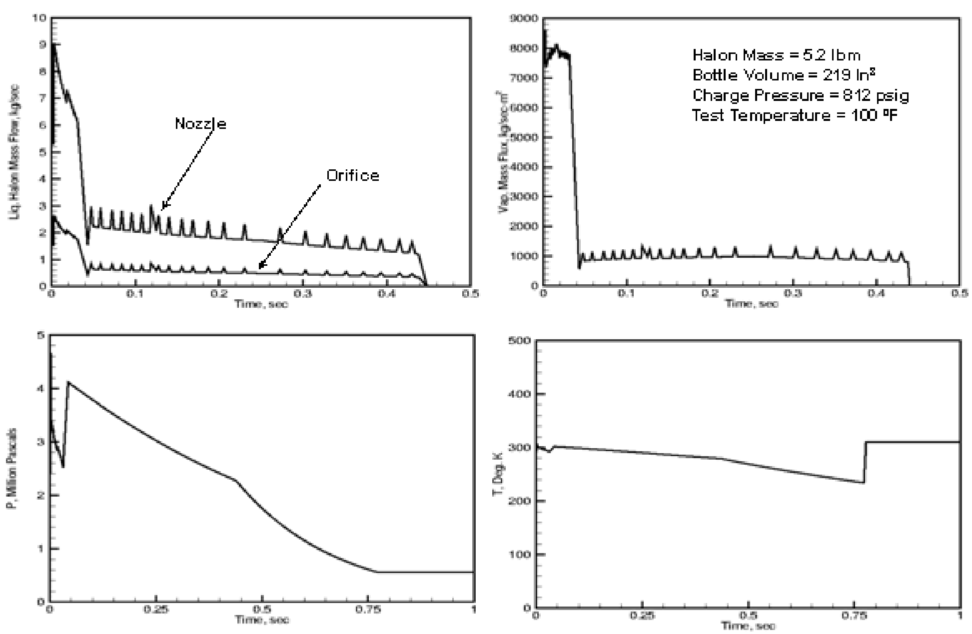


Figure 12: Hflowx Predicted Transient Two-Phase Jet Flow Properties at the Exits of the Injection Nozzle and the Orifice

The changes in concentration distribution inside the compartment during the entire fire suppression process can be better understood using the series of contour plots in figure 14. It is noted that, as shown by the plot at approximately 0.6 second in figure 14, the concentration of Halon 1301 propagates beyond the vented airflow inlet boundary for a very short period of time due to the strong momentum imparted from the four suppressant jets.

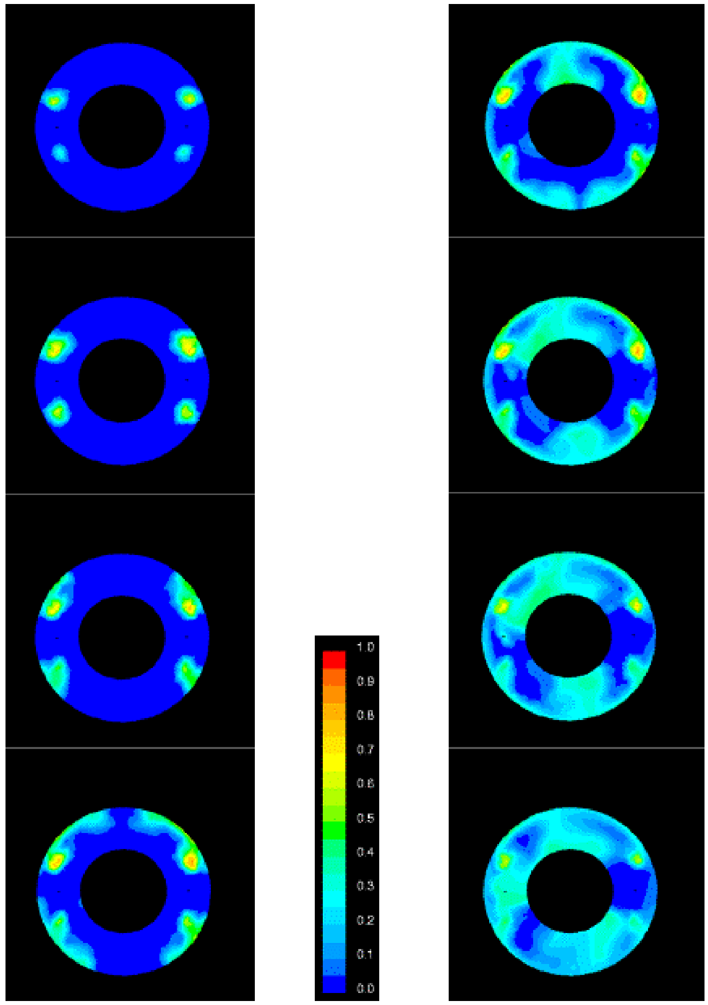


Figure 13: Front View of Jet Injection and Mixing with Counter-Flowing Air Stream (elapsed time increases from the top to the bottom, from the left to the right columns)

This trend indicates that the upstream boundary of the present solution domain should be extended farther upstream to contain the entire mixing/propagation inside the solution domain during the fire simulation process. In other words, the selection of the boundaries that define a solution domain of a vented compartment should be based on the two-phase flow phenomena during the entire fire suppression process.

Finally, the comparison of the predicted and the measured concentration histories at the 12 probe locations inside the Nacelle Fire Simulator compartment is shown in figure 15. The predicted concentration histories are in good correlation with the test data. Figure 15 also shows that the predicted concentration levels at the 12 different probe locations are, on the average, greater than 6% by volume for longer than a half-second period. The slight underprediction of the peak concentration levels was most likely due to the unsteadiness of the airflow at the inlet boundary during the injection period and the slight loss of suppressant caused by the propagation of concentration across the upstream boundary of the solution domain.

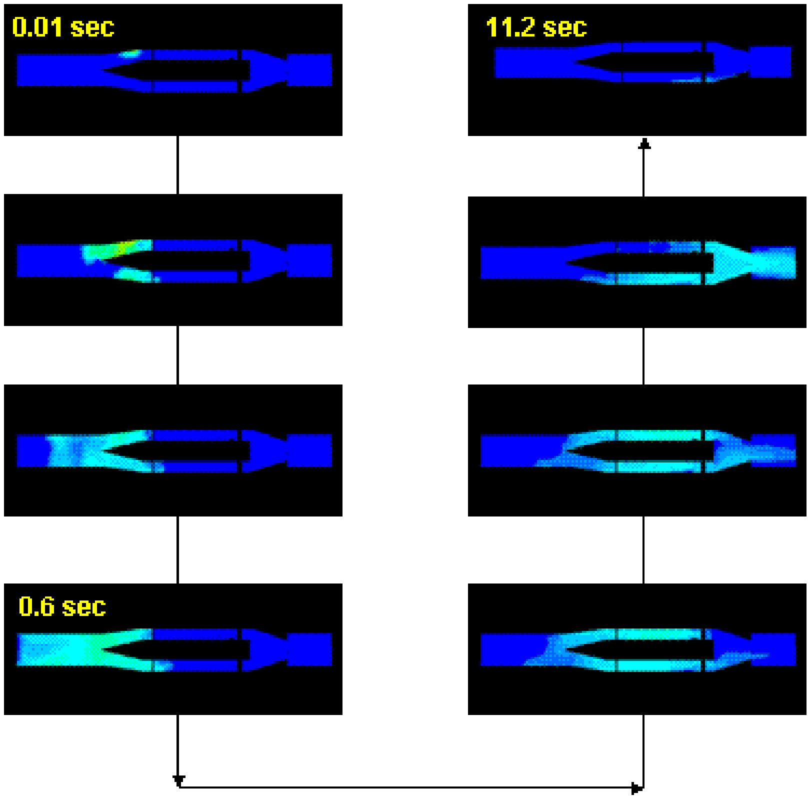


Figure 14: Side View of Jet Injection and Mixing With Counter-Flowing Air Stream

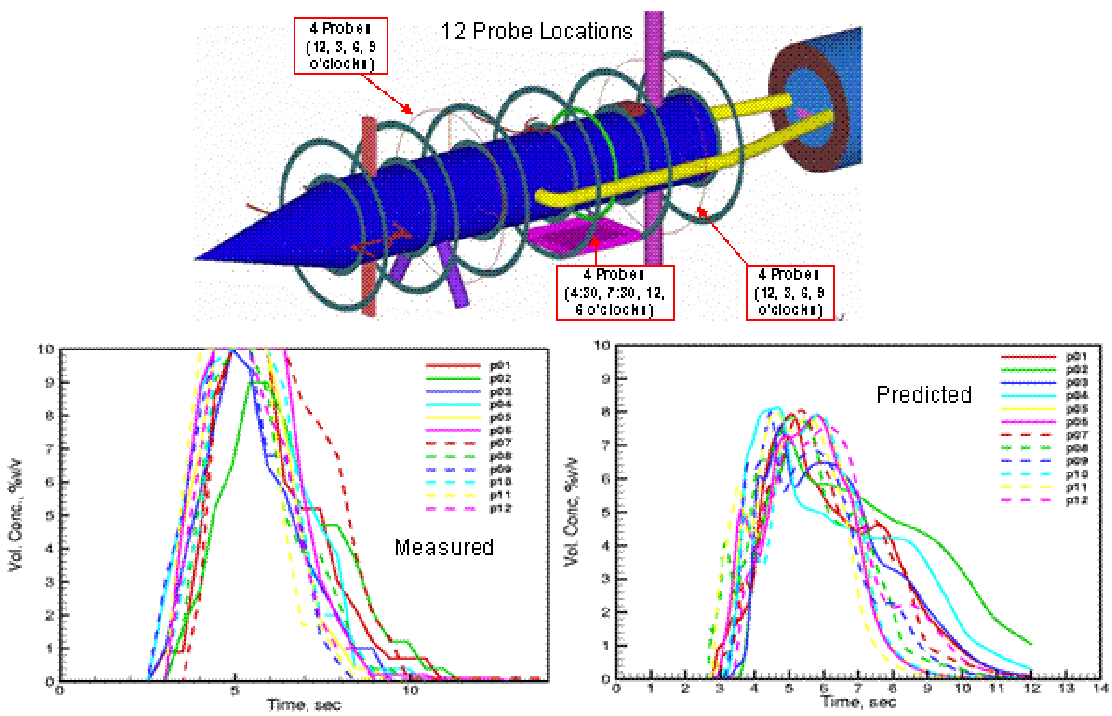


Figure 15: Comparison of Predicted and Measured Concentration Histories at 12 Probe Locations Inside the Nacelle Fire Simulator

APU Compartment Case

The flow conditions for the compartment cooling air and the engine exhaust gas during the steady-state APU operation considered are shown in table 3. Again, the flow and temperature distributions in the APU compartment and the exhaust pipe were predicted to obtain the initial flow conditions in the unsteady analysis of the flow inside the APU during the engine shutdown. The engine exhaust gas was treated as air with elevated temperature.

Table 3: Airflow and Engine Exhaust Conditions

Items	Conditions
Steady-state APU inlet airflow	12.8 lbm/s
Ambient air temperature	79 °F
Ambient static pressure	14.7 psia
Steady-state engine exhaust gas flow conditions	10.15 lbm/s, 980 °F

In the certification tests of the APU, the APU shut down from steady-state operation and there was a 3-sec waiting period until the agent was injected into the compartment. This implies that the exhaust flow rate decreases with time due to the reduction of APU shaft speed during the engine shutdown ($0 < t < 3$ sec) and the fire suppression period ($3 < t < 15$ sec). For this transient airflow reduction due to engine shutdown, a steady-state airflow analysis was conducted first. Next, using the steady-state airflow distribution as the initial condition, the transient analysis of airflow during the initial 3 sec of the engine shutdown period was conducted to obtain accurate initial airflow distribution before the agent injection.

Figure 16 shows the predicted initial airflow distributions at the air inlet duct and inside the APU compartment before agent was injected. Defining the solution domain to include the outside of the APU and the inside of the exhaust duct allowed the prediction of the detailed distribution of vented airflow inside the APU compartment.

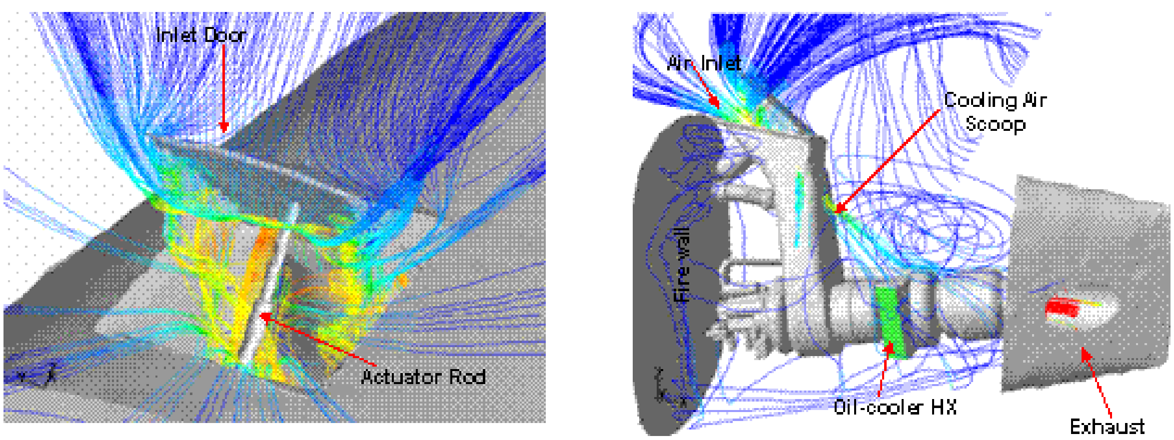


Figure 16: Predicted Path Lines of Cooling Air Inside the APU Compartment

Next, Hflowx was used to predict the agent jet flow boundary conditions at the interface between the injection nozzle exit and the APU compartment. The storage bottle charge and the discharge conditions of Halon 1301 that were used as the input data in the Hflowx analysis are listed in table 4. Figure 17 illustrates the predicted results: the static pressure, the static temperature, the liquid-phase mass flow rate, and the vapor-phase mass flux at the injection nozzle exit.

Table 4: Storage Bottle and Discharge Conditions

Items	Conditions
Total injected mass	14 lbm
Storage bottle volume	536 in^3
Bottle charge	600 psia, 70 °F
Discharge temperature	70 °F
Injection nozzle diameter	1.0 inch
APU compartment	14.9 psia, 95 °F

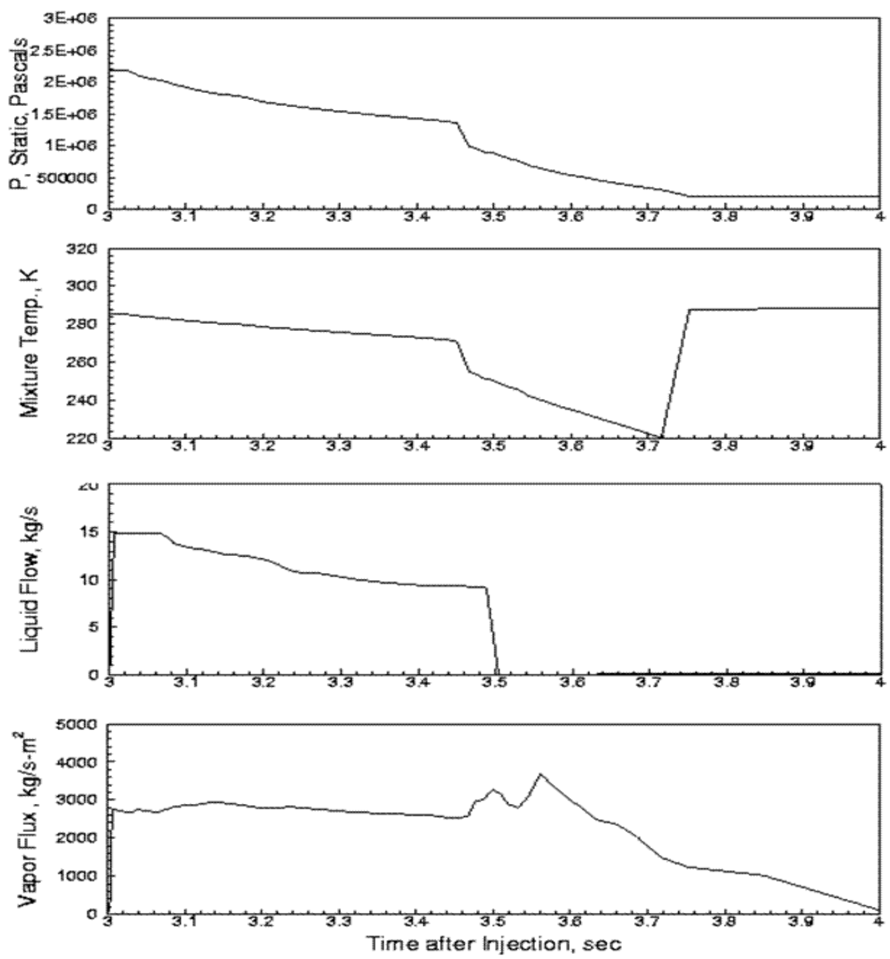
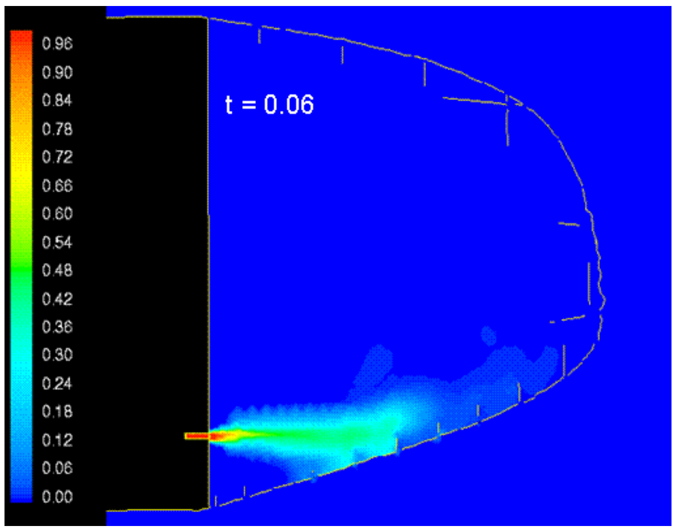


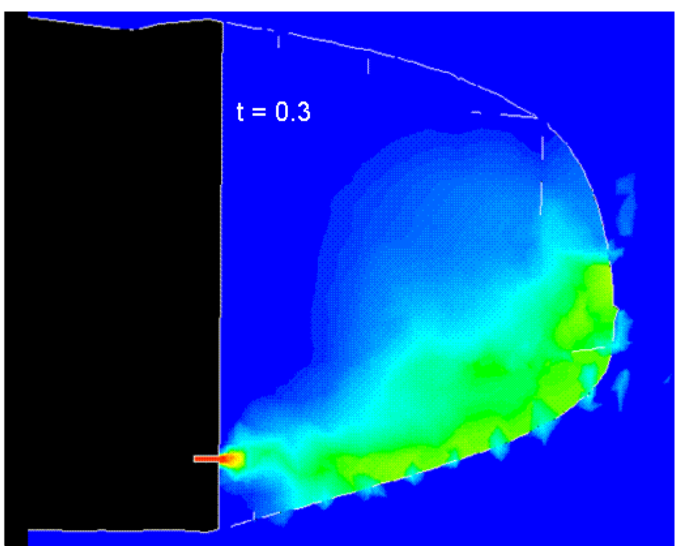
Figure 17: Hflowx Predicted Transient Two-Phase Jet Flow Properties at the Injection Nozzle Exit

In the transient analysis of the agent injection process, the initial size, velocity, temperature, and total mass flow rate of the droplets should be specified as the additional boundary conditions at the nozzle exits. In the present simulations, an estimated droplet size of 100 microns was used. The droplets were allowed to break up by the aerodynamic interaction with the mixture gas into smaller sizes using a break-up model in the chosen CFD code. The initial velocity and the initial temperature of the droplets were calculated based on the Hflowx predictions: the liquid-phase mass flow rate, the mass fraction of liquid phase, the nozzle area, and the density.

Figure 18 shows the predicted concentration distributions of Halon 1301 in the vertical plane through the nozzle during the agent injection. Figures 18a and 18b clearly illustrate how the Halon 1301 concentration penetrates and propagates inside the APU compartment with time.



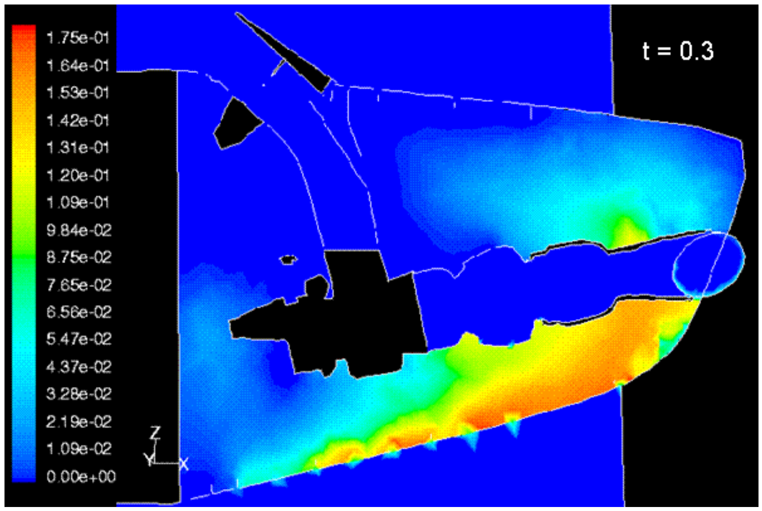
a) at 0.06 second after the Injection



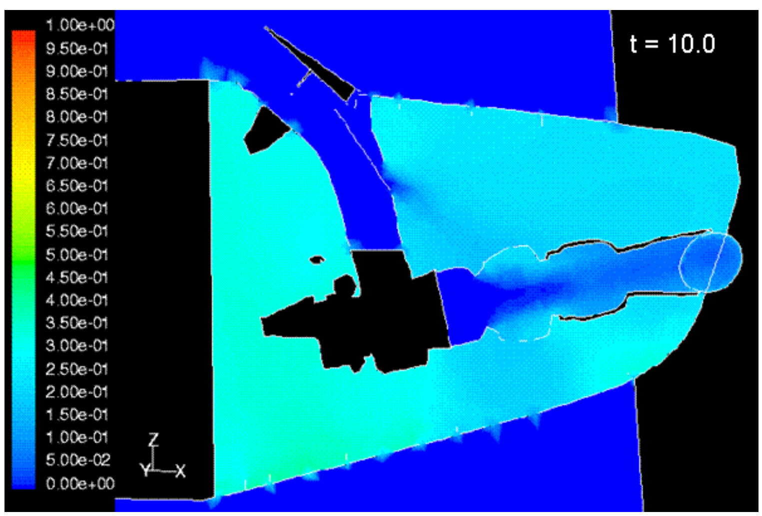
b) at 0.3 second after the Injection

Figure 18: Temporal 2D Concentration Distribution of Halon 1301 Inside the APU Compartment (vertical plane through the injector)

The concentration distributions in the vertical planes through the air inlet duct, the APU engine, the exhaust gas pipe, and the exhaust exit are shown in figures 19a and 19b at 0.3 and 10 sec after the agent injection, respectively. Figure 19a shows the density stratification at the bottom of the APU case and the Halon-free airflow inside the exhaust pipe. However, figure 19b shows, at 10 sec after the agent injection, the nearly uniform concentration distribution everywhere inside the compartment including inside of the exhaust pipe.



a) at 0.3 second after the Injection



b) at 10.0 seconds after the Injection

Figure 19: Temporal 2D Concentration Distribution of Halon 1301 Inside the APU Compartment (vertical planes through air inlet, engine, and exhaust pipe)

The measured and the predicted concentration histories at the 12 different concentration probe locations (figure 2) are compared in figure 20. Comparison of the two results reveals that the present simulation method predicted concentration history trends that were similar to the measured data, even though some concentration levels during the concentration propagation period were under-predicted 3% to 5% by volume. The causes of the under-predicted concentration levels (i.e., faster dilution of concentration) may be attributed to the combination of several sources of uncertainty in the simulation, such as the transient airflow boundary conditions, the pressure drop characteristics of the air and agent gas mixture across the oil-cooler heat exchanger, or the initial conditions of the droplets of liquid-phase agent. The predicted results in the present analysis are, however, very encouraging because the flow physics of the entire fire suppression process - the agent injection and the concentration propagation - inside the complicated APU compartment could be captured reasonably well.

The method is presently under improvement to enhance the prediction accuracy for (1) applications in new firex system designs for new airplanes and (2) the system changes to accommodate future replacement agents.

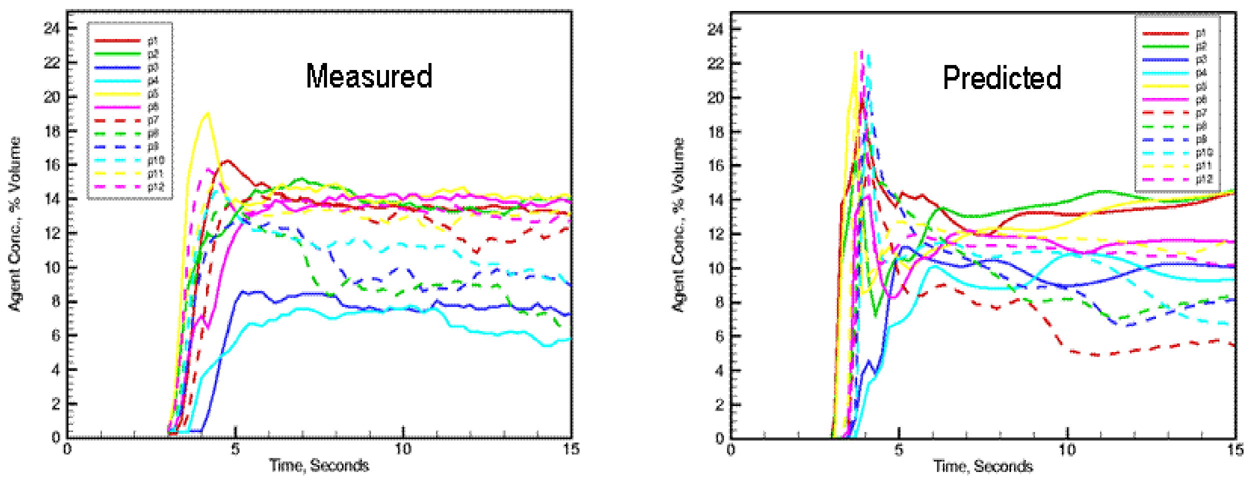


Figure 20: Comparisons of Measured and Predicted Concentration Histories

Analysis Cycle Times

Table 5 shows the approximate elapsed times required for each analysis when CFD analyses were conducted in parallel computation mode. The total time for simulating one case is approximately a week using the limited number of CPUs. Obviously, the analysis time can be further reduced with the increase of computers. The required analysis time of the present method depends on such things as (1) the total simulation time, (2) the size of the CFD mesh, (3) the number of injection nozzles, (4) the number of droplet sizes, (5) the number of droplet starting locations per nozzle, (6) the number of computer processors, and (7) the convergence criteria.

Table 5: Analysis Types and Required Times

Analysis Types	Computer Platform	Analysis Time	Remarks
Firex system	SGI Octane2, 400 MHz	<1 min	
Steady-state analysis of initial airflow distribution	ORIGIN 3800 (4 CPUs)	~0.5 day	0.32 Mcells
Unsteady analysis of agent injection and concentration propagation	ORIGIN 3800 (6 CPUs)	~1 week	APU with 1 injection nozzle

Key Factors for Improved Simulation

Some factors that are important in improving the accuracy of simulation have been identified while developing the simulation method. They can be summarized as follows:

- Selection of a solution domain based on the entire fire suppression process.
- Advanced flow physics models for the two-phase agent jet from the injection nozzle.
- Accurate vented airflow data.
- Accurate thermodynamic properties of the liquid and the vapor phases of agents.

CONCLUSIONS

Based on the simulations conducted so far, the following conclusions can be drawn:

- A computer simulation method has been developed for the fire suppression process inside engine core and APU compartments.
- The capabilities of the methods have been demonstrated by simulating the firex tests of the Nacelle Fire Simulator and the APU.
- The predicted concentration histories correlate well with the measured data.
- The simulation methods need to be improved for more accurate prediction of concentration histories.

ACKNOWLEDGMENT

The author thanks D. Ingerson of the FAA, W. J. Hughes Technical Center who provided firex test data for the validation of the simulation method. Thanks also go to many people at The Boeing Company for supporting the work: C. Roseburg for thermodynamic properties of agents, D. Lackas and J. Petkus for certification test data, M. Dunn and D. Dummeyer for engine cooling airflow data, and M. Grueneis and R. Moody for mesh generation.

REFERENCES

1. FAA Advisory Circular AC No. 20-100, “General Guidelines for Measuring Fire-Extinguishing Agent Concentration in Powerplant Compartments,” September 21, 1977.
2. Elliot, D. G., et al., “Flow of Nitrogen-Pressurized Halon in Fire Extinguishing Systems.” JPL Publication 84-62, Nov. 1, 1984.
3. ICEM CFD Engineering, ICEM-CFD Tutorial Manual: Meshing Modules, version 4.0, August 1999.
4. Wilcox, D. C., Turbulence Modeling for CFD, the DCW Industries, 1998.
5. Fluent, Incorporated, FLUENT 6 - User's Guide, Jan. 2002.
6. Crowe, C. T.; Trott, T.R.; and Chung, J. N., “Numerical Models for Two-Phase Turbulent Flows,” *Ann. Rev. Fluid Mech.* 28, 11-43.