SANDIA REPORT

SAND96-2714 • UC-700
Unlimited Release
Printed November 1996

Computational Fire Modeling for Aircraft Fire Research

V. F. Nicolette

Prepared by
Sandia National Laboratories
Albuquerque, New Mexico 87185 and Livermore, California 94550
for the United States Department of Energy
under Contract DE-AC04-94AL85000

SF2900Q(8-81)

Issued by Sandia National Laboratories, operated for the United States Department of Energy by Sandia Corporation.

Printed in the United States of America. This report has been reproduced directly from the best available copy.

Available to DOE and DOE contractors from
Office of Scientific and Technical Information
PO Box 62
Oak Ridge, TN 37831

Prices available from (615) 576-8401, FTS 626-8401

Available to the public from
National Technical Information Service
US Department of Commerce
5285 Port Royal Rd
Springfield, VA 22161

NTIS price codes
Printed copy: A05
Microfiche copy: A01

Distribution
Category UC-700

SAND96-2714
Unlimited Release
Printed November 1996

Computational Fire Modeling for Aircraft Fire Research

V. F. Nicolette

Unsteady and Reactive Fluid Mechanics
Sandia National Laboratories
Albuquerque NM 87185-0836

Abstract Follows

Abstract

This report summarizes work performed by Sandia National Laboratories for the Federal Aviation Administration in support of IA DTFA03-93-X-00014. The technical issues involved in fire modeling for aircraft fire research are identified, as well as computational fire tools for addressing those issues, and the research which is needed to advance those tools in order to address long-range needs. Fire field models are briefly reviewed, and the VULCAN model is selected for further evaluation. Calculations are performed with VULCAN to demonstrate its applicability to aircraft fire problems, and also to gain insight into the complex problem of fires involving aircraft.

Simulations are conducted to investigate the influence of fire on an aircraft in a cross-wind. The interaction of the fuselage, wind, fire, and ground plane is investigated. Calculations are also performed utilizing a large eddy simulation (LES) capability to describe the large-scale turbulence instead of the more common k-ε turbulence model. Additional simulations are performed to investigate the static pressure and velocity distributions around a fuselage in a cross-wind, with and without fire. The results of these simulations provide qualitative insight into the complex interaction of a fuselage, fire, wind, and ground plane. Reasonable quantitative agreement is obtained in the few cases for which data or other modeling results exist.

Finally, VULCAN is used to quantify the impact of simplifying assumptions inherent in a risk assessment compatible fire model developed for open pool fire environments. The assumptions are seen to be of minor importance for the particular problem analyzed. This work demonstrates the utility of using a fire field model for assessing the limitations of simplified fire models.

In conclusion, calculations performed as part of this work successfully demonstrate the significant potential that fire modeling has for aircraft fire research. The application of computational fire modeling tools herein provides both qualitative and quantitative insights into the complex problem of aircraft in fires.

Table of Contents

List of Figures

Acknowledgments

This work was supported in part by the Federal Aviation Administration Technical Center, Atlantic City International Airport, New Jersey. The author wishes to thank Dr. Thor Eklund (FAATC) and colleagues for their interest and helpful discussions regarding this work.

Portions of this work were sponsored by other agencies, including the Defense Nuclear Agency and the Sandia National Laboratories Engineering Sciences Research Foundation.

The author gratefully acknowledges the contributions of other Sandia National Laboratories Fire Science and Technology Team members to this work, particularly Louis Gritzo, Jaime Moya, Russ Skocypec, and Sheldon Tieszen.

This work was conducted by Sandia National Laboratories which is operated by Lockheed-Martin Corp. for the U.S. Department of Energy under Contract DE-AC04-94AL85000.

Intentionally Left Blank

1.0 Introduction

The purpose of this report is to document and summarize part of the work performed in support of IA DTFA03-93-X-00014*, an Interagency Agreement between Sandia National Laboratories (Sandia) and the Federal Aviation Administration (FAA). Other work performed under this agreement can be found in Tieszen, et al. (1996). As part of this agreement Sandia performed work for FAA in the area of fire modeling for aircraft fire research.

As part of the Aviation Safety Act of 1988 (Public Law 100-591), the FAA was mandated to perform long-range research that will lead to new technologies for improving aircraft fire safety. It is important to note that this research was specifically directed to be long-range in nature (with a consequently long time frame for implementation and payback), and not directed toward aircraft fire safety issues that are more short-term in nature. In response to this mandate, the FAA created a Fire Research Branch and developed a Fire Research Program Plan (Eklund (1993)). The stated goal of the Fire Research Program is to eliminate fire as a cause of fatalities in aircraft accidents. The following objectives have been identified to achieve this goal:

- determine where counter measures can be most effective
- determine what countermeasures have to do to be most effective
- determine what performance requirements would make interior materials virtually flame-proof.

Six important fire research areas (or thrust areas) were identified in order to meet these objectives. These were:

- Fire Modeling
- Vulnerability Analysis
- Fire Resistant Materials
- Improved Systems
- Advanced Suppression
- Fuel Safety

In Eklund (1993), fire modeling was determined to be an important area for research, since the wide range of conditions associated with aircraft fires precludes experimental determination of all important parameters. Models were deemed necessary to predict the dynamic interaction of the fire and fuselage, as well as to provide tools for analyzing risk and vulnerability. Fire models were also acknowledged to allow for analysis of the effectiveness of improvements to aircraft fire safety, and for determination of when such improvements reach the point of diminishing returns.

*This work was performed during the author's assignment at the FAATC as a Visiting Scientist.

This report summarizes work performed as part of the Fire Modeling thrust during the period 1993-1996. There were two main aspects to this work: 1) develop a program plan for the Fire Modeling thrust area, and 2) initiate research as identified in the program plan. Both aspects of the work are discussed herein. Section 2 of this report identifies the technical issues associated with aircraft fire modeling research, and technical methods for addressing these issues. Reference is also made to a research program plan that was drafted for the Fire Modeling thrust area. Due to a lack of funding, the plan was never finalized. It is included here (in the form of an appendix) to provide a framework for understanding the modeling research that was initiated as part of this effort.

Some of the research tasks identified in the draft program plan were initiated (as funding allowed). The types of fire models that were deemed most appropriate for aircraft fire research were identified. Existing fire field models were reviewed, and the best two selected for further evaluation (as described in Section 3). Of these two, one model (described in Section 4) was evaluated for its technical capabilities and applicability to the problem of aircraft fire simulation.

Section 5 documents the results of calculations that were performed to investigate the influence of fires on aircraft fuselages. The results include: 1) a demonstration calculation of an aircraft in a fire with a cross-wind, 2) a calculation of a mock fuselage in a pool fire with a cross-wind, 3) a demonstration of a large-eddy simulation capability for a fire involving a mock fuselage, and 4) the calculation of the static pressure distribution around a mock fuselage in a cross-wind, with and without fire. Another type of fire model, a risk-assessment-compatible fire model (or RACFM) was also evaluated as part of this work. Section 6 documents an evaluation of the accuracy of the RACFM results, and also assesses the validity of the underlying assumptions of the model.

2.0 Fire Modeling Thrust Program Development

As part of the FAA Fire Modeling thrust, the development of a structured research plan was initiated. The first step in this development was the identification of the technical work (in a strategic sense) that was needed. The technical issues relevant to aircraft fire modeling research were identified, along with technical methods for addressing those issues. Fire scenarios of importance to aircraft safety were identified: external pool fires, internal fires resulting from (and coupled to) an external pool fire, and internal in-flight fires. The various types of fire models were assessed for their applicability to aircraft fire research. Modeling questions requiring investigation were identified for both external and internal fires involving aircraft. The need to investigate and quantify model uncertainty and sensitivity was highlighted, as well as the need to extensively validate fire models against appropriate experimental data.

2.1 Fire Scenarios of Interest for Aircraft

There are basically two types of fire scenarios that are of interest for the FAA Fire Research Program: fires originating internal to the fuselage pressure hull, and fires originating external to the fuselage. These two types of fires are very different from each other due to geometry, ventilation, and fuel considerations. Internal fires can result from an in-flight fire, or a fire that ignites when the plane is on the ground. The focus of internal fire modeling is on the effects on critical systems, the internal cabin environment, and its effects on habitability of the cabin. External fires are generally a post-crash concern following the dispersion of fuel, although there have been external fires in-flight (power plant and landing gear fires). The focus of external fire modeling is on the protection which the fuselage hull provides to the passengers inside. The above separation of aircraft fires into external and internal fires is somewhat artificial. In actual post-crash fire environments with the fuselage intact, the external fire environment is important initially, followed by the internal cabin environment once the fuselage has begun to melt through, or internal materials begin to decompose and release toxic gases and smoke, or the interior materials are actually ignited. The two fire environments are actually coupled together once fuselage burn-through has begun, or if wind effects lead to fuselage ingestion of combustion products from an external fire.

2.2 Fire Models of Interest for Aircraft

Traditionally, there are basically two different types of fire models that can be applied to model aircraft fires: field models and zone models. Also, a few material burning models have been developed to describe the thermal response and burning of aircraft materials. The particular models applied depend on the scenario of interest and the information desired from the model.

- **Field Models:** Field models are based on a solution of the governing partial differential equations, and hence are the most robust of the fire models. They are robust in the sense that they use a first-principles approach with a minimal reliance on experimental correlations, and therefore can be applied to a wide variety of fire scenarios (even those for which there are no experimental data), including both external and internal aircraft fires. A field model is best suited to giving very detailed information about the fire environment, including the spatial and temporal variation of the temperatures, heat fluxes, velocities, and species concentrations. Field models usually require significant computational power.

- **Zone Models:** Zone models use empirical models and correlations to predict the consequences of a fire on a compartment environment. Zone models lump geometric information into each zone of the model. They are generally not robust, in that the correlations upon which they are based only apply to very specific geometries and ventilations. Zone models are best suited to providing temporal and spatially averaged outputs for simple geometries and ventilation conditions, such as building fires. Plume models can be viewed as a subset of zone models, as they provide information about the region just above the combustion zone. Plume models generally have some simplified representation of the spatial distribution within the plume. Historically, plume models have been applied to model internal fires just above the combustion zone, or an external fire without any large object in it and without wind effects.

- **Material Burning Models:** Material burning models provide a method for determination of the heat release rates of various fuels in various configurations. They are therefore important models for internal fires, where the heat release rate, flame spread, and ignition of fuel can drive the compartment fire environment. Heat transfer through some burning materials (such as rigid foams) is not well understood, as it is difficult to even estimate what the thermal transport properties should be. Generally, material models are used by themselves to investigate the relative fire behavior (e.g., heat release rate, flame spread) of various materials. To date, there has been only limited coupling of material models to field and zone models, and what has been done is very crude (i.e., using quasi-steady burning models in transient fire models, such as is done in the COMPRN model). This coupling is quite important, since the fire environment greatly influences the rates of material devolatilization and burning. The thermal response of composite materials is also an area where much research needs to be done (due to anisotropic conduction), especially as composites begin to play a larger role in aircraft materials.

Any or all of the above models can be used as part of a fire risk and vulnerability analysis. Since fire risk assessment generally requires the investigation of a large number of fire scenarios, computer resources may be a concern. This points out the importance of high performance computing. It may also be advantageous to streamline a given model to efficiently calculate only a specific scenario set, thereby reducing the required computation time. This actually leads us to another class of fire models, that are different in approach than the others: risk assessment-compatible fire models (RACFM).

- **RACFM Models:** RACFM are derived from the more traditional fire models described above by streamlining them for a specific fire problem or type of scenario. The model used will depend on the scenario of interest. The distinction between the RACFM and the traditional models is that the RACFM focus on only the dominant physics of the scenario. RACFM can incorporate elements of zone, field, and/or materials models, but only if they capture the dominant physics of the scenario. Unlike field models, RACFM contain models for only the dominant physical processes, and discretize the field only where it is necessary. Unlike zone models, RACFM have all of the important physics plus the requisite spatial discretization required to provide accurate solutions for a scenario of interest. Because they do not carry any excess baggage, but only model the essential physical processes, RACFM are optimal models for risk assessment applications, providing a balance between the physics and CPU time. An example of a RACFM would be the Sandia Gray Gas Model, described more fully in Section 6.

2.3 Technical Questions Concerning the Modeling of Aircraft Fires

At first glance, there appear to be many different technical questions concerning the modeling of aircraft fires that should be investigated by a long-range research program. A preliminary list has been identified below for both external and internal aircraft fires. Internal fires have been further divided into fires which are coupled to fuel fires, and fires which are not (generally in-flight fires, although they may occur during taxi). The following list has been generated with regard to: 1) the technical questions regarding the scenarios that one would want to investigate using a fire model, 2) the technical questions regarding the application of existing fire models to scenarios of interest, and 3) the technical questions regarding the development of new, extensively validated fire models for aircraft fires.

Modeling Questions: External Fires

- In-flight fires: It may be of interest to investigate the effects of in-flight fires EXTERNAL to the cabin. A fire external to the pressure hull could conceivably damage enough other critical components to cause an aircraft to crash. Modeling of this environment would present many unique challenges because of the high velocity, low temperature, and low pressure ambient conditions.

- Fuel dispersion: The dispersion of fuel upon impact (and following impact) is an important variable in determining the resultant fire environment. Generally, fire models require (as an input) an estimation of the spilled fuel distribution area and thickness. Wind effects must also be included in the modeling of dispersion.

- Fuselage integrity post-crash: Fire models also require some assumptions on the fuselage structure following impact (again, this is an input to a fire model). Crash dynamics analysis can be used to determine a range of possible fuselage damage states, for which fire modeling can then be performed. Such an analysis could determine whether the fuselage is completely intact following landing/impact (and thereby necessitating the modeling of an external fire until burn-through is reached), or whether the fuselage has been damaged to the extent that the internal environment in the cabin should be immediately coupled to the external pool fire environment.

- Initial Transients: The initial transient period of a post-crash fire consists of impact, dispersion, ignition, fireball, and flame spread across the spilled fuel. Some of these have been discussed above, but the remaining issues (ignition, fireball, and flame spread) are of importance.

- Pool effects: The heat transfer from the flame volume to the liquid fuel on or wetting the ground (in the case of a pool or spill fire) vaporizes the fuel and thus drives the combustion process following ignition. There are many elements of the pool behavior that are poorly understood, such as: the absorption of thermal radiation as a function of depth of the pool, absorption of thermal radiation by the vaporized fuel immediately above the pool surface, and the influence of convective effects in the pool on pool surface temperature. Since this area is poorly understood, yet drives the combustion process, research into it could yield significant benefit.

- Wind effects: Large vortical structures can exist that heavily influence the local temperature and velocity fields, and thus the local heat fluxes to the fuselage. Additionally, wind effects can generally lead both to highly time-dependent behavior of the fire environment, and varying spatial heat fluxes. The ability of a field model to accurately capture wind effects over a fuselage appears to have great potential, but has not been proven. Time-dependent boundary conditions should be investigated for the role they play on the fire environment.

- Ventilation into cabin: Ventilation paths from the external fire environment into the cabin should be further investigated. This has the potential to introduce combustion products and heat into the internal environment before fuselage burn-through is achieved. This ventilation also has a great dependence on wind conditions.

- Fuselage burn-through: Apart from the determination of fuselage damage upon impact, a fire model capable of modeling the burn-through of the fuselage must be developed. This will require the incorporation of burn-through models for aircraft fuselage materials, and perhaps modeling of such details as the aircraft frame and insulation locations. Additionally, the capability must be developed to allow a fire model to suddenly shift from modeling only the external pool fire environment, to modeling both the external and internal environments.

- Field model issues: While some of the issues above apply directly to existing fire field models (and the development of new ones), there remain basic problems with present-day field models. The major technical questions in need of further research appear to be:

- the details of the combustion modeling (in particular, how the turbulence model couples to the combustion process)
- the turbulence model (apart from its coupling to the combustion model) and its impact on local flame temperatures and soot concentrations, and the importance of capturing large-structure effects (vortices) with the model
- the soot model (including the generation, agglomeration, radiation properties, and combustion of soot particles), and its sensitivity to soot concentration, composition, and size
- the calculation of radiative heat transfer, including the combustion products' radiative properties, and non-homogeneous temperatures in a control volume
- the calculation of convective heat transfer to large objects in the fire, including the validity of the law of the wall approach

- Zone model issues: It is doubtful that zone models can be effectively applied to study external fires of interest to aircraft, because of the spatial averaging over each zone. For an aircraft in a fire, large variations in the thermal, velocity, and concentration fields lead to very large variations in the local radiative and convective heat fluxes. The ability to capture this variation with a zone model does not exist.

- RACFM issues: Although they are derivatives of field or zone models, RACFM usually incorporate a large quantity of experimental data to account for phenomena that are not directly modeled. Therefore, the range of validity of RACFM must be thoroughly understood. Some of the major technical questions regarding RACFM for aircraft fire modeling appear to be:
  - The incorporation of wind effects into the model, both in terms of direction relative to the fuselage and fluctuations in magnitude, should be included.
  - Since RACFM usually model only a portion of the fire field, appropriate far-field boundary conditions must be identified and coupled to the RACFM.
  - Experimental data is needed that is sometimes difficult to obtain, such as a combustion source term or soot absorption coefficient.
  - A method to assess what the dominant physical processes are for a specific scenario is needed.
  - Coupled with the above, a method for assessing the validity of assumptions in the RACFM.

- Computing power: The need for high performance computing power is clear in view of the magnitude of the problem under study. This is true for internal fires as well. High performance computing (such as supercomputers, massively parallel, and distributed parallel systems) is essential for field models and also for RACFM, since risk assessments generally require investigation of a large number of scenarios.

- Model uncertainty: This issue applies to fire models in general, for both internal and external fires. Because of its importance, it will be discussed later, following the issues concerning the modeling of internal fires.

The above list is not intended to be exhaustive in scope, but rather representative of some of the technical questions toward which long-range research should be directed for the modeling of external aircraft fires. A corresponding list of questions for the modeling of internal aircraft fires is as follows:

Modeling Questions: Internal Fires Coupled to Fuel Fires (Post-Crash)

- Material burning rate: The heat release rate of burning cabin materials drives much of the internal cabin fire environment. Models for the heat release rate as a function of time for charring and multi-layered materials need to be developed. The local fire environment conditions (velocities, temperatures, oxygen) are critical inputs to material burning models.

- Flame spread: Flame spread models must be developed for typical materials burning under real fire conditions. Generally, flame spread models should include local fire environment conditions (velocities, temperatures, oxygen, and soot concentrations), as they are heavily influence by local convective and radiative heat transfer. Also, orientation of the material is an important parameter.

- Ventilation: Post-crash ventilation is affected by wind. The potential to pull fire products from an external fire into the cabin should be investigated, as well as possible interactions between the ventilation and external fire environment once the fuselage has begun to melt through. For internal fires, the effect of local ventilation is critical for determining such things as air entrainment into the flame volume, flame merging and tilting, dispersion of combustion products, and flame spread along combustible materials.

- Opening of doors: This could have been included under the ventilation issue, but has been separated out to clarify the issue. Once a fire has developed in an aircraft cabin, the opening of a door will greatly change the fire behavior. When a door is opened (such as to provide an escape path), fresh air may enter the cabin. This has at least two effects: it further pushes the combustion products up near the ceiling, and may provide more oxygenated air to the burning materials. The opening of a door will also influence all of the local ventilation, and thereby that of the entire compartment.

- Fuselage burn-through: See the above discussion for external fires.

- Internal auto-ignition: It is plausible that high-strength titanium hulls may one day be used for fuselages. The melting point for titanium (1953 Kelvin) is greater than the maximum flame temperatures that can be expected for hydrocarbon fuels. Therefore, melting of the fuselage may become a non-issue if titanium hulls are used. However, if titanium hulls are employed, the issue arises that the internal materials could auto-ignite from an external fire source just by heat transport through the wall of the fuselage.

- Toxic gas production: For internal fires, the production of combustion products such as CO and smoke becomes an important factor. This production is generally a strong function of the local combustion conditions, as well as the type of combustible material. Local ventilation may influence the rate of production of toxic gases, as well as the rate of dispersal throughout the fuselage.

- Visibility: The ability to evacuate a fuselage is strongly influenced by the visibility of exit signs, escape lighting, and the degree to which the combustion products cause eye irritation and tears. Soot production and distribution is an important factor in determining visibility in the fuselage during a fire (along with acid gas generation). Soot production is generally not well understood, especially for the conditions representative of a fire environment.

- Field model questions: There are a few more field model questions (in addition to those noted under the external fire category above) relative to internal (post-crash) fire modeling:
  - Radiative and convective heat transfer from the hot gases to surfaces becomes the dominant heat sink for internal fires. Tests at FAA and Sandia indicate that 80-90% of the energy released in an internal fire is deposited in the walls and ceiling surfaces. Models of these processes in a field model will have to be very accurate to capture temperature and velocity fields.
  - Very tight coupling to materials models becomes essential, as noted above.
  - Soot deposition on (and charring of) surfaces changes the radiation characteristics, and may be important to include in the model.
  - The transport and physical properties for hot combustion products may differ significantly from properties at ambient conditions. The sensitivity of these properties to fire temperatures and compositions should be explored.
  - The production of toxic gases (such as CO) becomes very important in the internal cabin fire. Models for the production, dissipation, and transport of each species must be included in the field model.
  - The effect of passengers and passenger seats in the cabin is also an important consideration for a field model, including the volume of free space, oxygen uptake, and induced motion.

- Zone model questions: Zone models have been most successfully applied to enclosures with relatively cubic geometries, like rooms and warehouses. The questions regarding using zone models for internal fires include those for a field model, plus several more. For example:
  - Fire entrainment correlations for fires in small, confined, complex geometries with complex ventilation.
  - Heat release rate correlations for fires in small, confined, complex geometries with complex ventilation, including the effect of vitiated atmosphere filled with combustion products.
  - Smoke release rate correlations for these same fires and conditions.
  - Because zone models, by their very nature, yield averaged quantities over a relatively large zone, they cannot be effectively applied to problems where local

information is desired and critical to the combustion process. Critical questions for zone modeling of internal aircraft fires include the ability to account for *local* velocity, *local* temperature, *local* oxygen, *local* soot, and *local* heat transfer effects. In view of the complex geometries, ventilation patterns, and fuel distributions inside most aircraft cabins, accounting for *local* effects is a formidable problem for zone modeling.

- RACFM questions: Because of the importance of complex geometries and ventilation effects in internal fires, RACFM will most likely be based on field models, although it is conceivable that certain elements of a zone model (such as DACFIR, (Reeves and MacArthur (1976)) may be incorporated. Basically, all of the questions regarding the field and zone models may also apply to the RACFM, depending upon the elements of each model that are incorporated into them.

- Computing power: The same comments as made above regarding external fires apply here.

- Model uncertainty: See the discussion on uncertainty/sensitivity/validation which appears later.

Technical Questions: Internal Fires In-Flight

Due to the relatively small incidence of internal in-flight fires, these fires were not deemed to be as high a priority for the development of the research program.

Technical Questions: Uncertainty/Sensitivity/Validation

Fire models are notoriously sensitive to the inputs and assumptions of the user, and yet, this strong influence of user judgment is very seldom discussed in the literature. All too often, fire models are only judged to be good or bad on the basis of a single (or a few) case(s), without any regard to the influence of different input parameters and assumptions on the results. Just showing that one set of input parameters produced reasonable results is often considered an accomplishment, without regard to the other cases that may have produced non-physical results. Unfortunately, someone using the same model with slightly different inputs can obtain completely different results.

In view of the above, there are some very important technical questions regarding model sensitivity and validation:

- Sensitivity to input parameters: This would include the impact of different input parameters upon the model results. How sensitive is the model output to changes in the fuel properties, the ventilation rate, wind conditions, material properties, heat transfer coefficients, entrainment coefficients, etc.? Are these sensitivities realistic? Do they reflect reality, or a weakness in the model?

- Sensitivity to modeling assumptions: This would include the impact of different modeling assumptions upon the model results. For example, how important is the assumption of thermal boundary conditions for the cabin materials? What influence does the particular grid and time step have on the results? How greatly are the combustion process and the local heat transfer rates affected by the choice of turbulence model employed? How important is the wind direction relative to the fuselage for a pool fire calculation? In general, both the assumptions that are inherent in a model as well as those that the user must make have to be evaluated for their impact on the model results. Assumptions inherent in a model that should be evaluated for impact on the solution uncertainty include: the order of solution of the equations, the method used to evaluate non-linear terms in the governing equations, the discretization formulation employed, etc.

- Overall sensitivity analysis to provide research direction: The above sensitivity issues are primarily aimed at understanding the effect of changes in individual parameters and modeling assumptions on the model results, with the intent of alerting the user to interpret the results accordingly. However, an overall sensitivity analysis could also be performed with a different goal: the direction of research efforts. Without some over-arching understanding of all of the sensitivities in a model, research efforts will be directed based on a best guess approach. However, if the effect on the results of the various input parameters and modeling assumptions could be evaluated systematically using something similar to a risk assessment methodology, then the areas in the model of greatest uncertainty (in terms of how they affect the model results) could be identified. Having identified these areas, the potential for reducing the largest areas of uncertainty could then be evaluated, and research directed down the paths shown to have the most impact on the model results. *This issue is one of the most foundational to a results-oriented long-range fire modeling research program, and may be the best framework from which to structure an overall approach to the fire modeling research.*

- Data for validation: Fire models should be extensively validated by comparison to experimental data. This helps determine some of the model sensitivities to both input parameters and assumptions, but does not usually catch all of them. Hence the need for the sensitivity studies mentioned above. For extensive validation to occur, a sufficient body of experimental data must exist. This data should span scenarios and parameters of interest, in order to assess the model over the widest possible set of conditions. Therefore, a major issue is the generation of experimental data applicable to aircraft fires over a wide range of conditions. The high costs associated with large- and full-scale fire testing will only allow a minimal number of tests to be performed. Since much data already exists that was not specifically derived for model validation, another major challenge would be to try and utilize such data in the model validation effort.

- Interpretation of data for validation: Along with the generation of experimental data applicable to aircraft fires, it must be emphasized that the quality of the data is important. This includes appropriate error estimates for the data, which are not straight-forward for large, sooty pool fires with high radiative heat fluxes. This would also include sufficient measurement of the important parameters affecting the experimental results (such as changes in the wind or boundary conditions, heating up of instrumentation with time, physical changes in structures with time). It is also recognized that instrumentation for a full-scale test can be significantly more complex than for small-scale tests (i.e. design of sensors, optical thickness problems, cooling requirements).

- Interpretation of model results for validation: Most fire models will calculate local gas temperatures, whereas experimentalists record thermocouple temperatures. Along with the previous issue of data interpretation, it would also be of great help to generate a thermocouple numerically in the fire model, to ease comparison with the experimental data. This could be done by post-processing the numerical results using a subgrid calculation to compensate for radiation heat fluxes, local gas velocities, and transient response. Similarly, other numerical instruments could be generated (for heat flux measurements, for example).

Modeling Issues: Integration with Materials Models and Risk Assessment Models

- Integration with materials models: As identified above, material flammability, heat release, flame spread, and fuel pool issues drive much of the fire environment. In turn, the local fire environment drives the response of the combustible materials. Because of this tight coupling (of non-linear phenomena, no less), it is critical that materials models be well-integrated into the long-range development of fire models. In general, it will not be enough to relate a time-averaged fire environment to the time-dependent materials model, or to relate a time-averaged material response (heat release rate, CO production, flame spread) to the time-dependent fire model. The coupling between the fire and materials models must be done on a time-dependent basis, if there is truly to be fidelity between the model and a real fire environment. The same comment applies to spatial averaging. Incorporating this coupling (in a rigorous sense) is a very ambitious goal.

- Integration with risk assessment models: The integration of fire modeling with risk assessment methodology is also critical. Risk assessment will determine the fire scenarios that are most probable, and thereby determine which scenarios should be investigated with a fire model. A fire model can then be used to determine the consequences of each particular fire scenario of interest, and thereby help in the quantification of the risk. It is therefore essential that fire models be developed and validated that can accurately model the scenarios of interest for the risk assessment. Fire models must also be integrated into the risk assessment effort in the sense that they must provide solutions in a reasonable amount of time, because a very large number of scenarios (~1000s) may be of interest for the risk assessment quantification.

2.4 Fire Modeling Thrust Program Plan

Based on the above work and the discussions which followed, a long-range research plan was drafted for the Fire Modeling thrust area. The preparation of this plan has not been completed as a result of a reduction/re-direction in funding. The last draft of the plan is included in this document as Appendix A, and the interested reader can find the details therein. The purpose of this plan was to define a tactical means for attaining the strategic goals of the Fire Modeling thrust area, which were:

- the development of analytical tools for aircraft fire modeling
- the integration of the other thrust areas into a coherent research program

The plan suggests 3 phases of activities for the Fire Modeling Thrust:

- Phase 1: research and development leading to a validated research code (see Appendix A for definitions) for modeling aircraft fires
- Phase 2: development of validated simulator tools and RACFM for use by the aircraft industry
- Phase 3: technical support and continued improvement of the simulator tools and RACFM during their application by aircraft designers/manufacturers/regulators.

The studies included in the remainder of this report were conducted as part of Phase 1 activities. They represent preliminary steps toward achieving the stated objectives of the Fire Modeling Thrust and the Fire Research Program.

Intentionally Left Blank

3.0 Evaluation of Fire Field Models

Fire modeling can play an important role in the design of fire-safe aircraft, and in providing computational tools for a quantitative (and/or probabilistic) assessment of aircraft fire safety. Research is necessary to advance the state-of-the-art in fire modeling to the point where fire models can be applied with confidence to aircraft fire safety analyses. From the discussion in Section 2, fire field modeling can be seen to have significant potential for application to aircraft fire problems. As a first step toward understanding the capabilities of existing fire models, an evaluation of existing fire field models for their applicability and development feasibility for aircraft fire modeling was undertaken. Note that the work documented in this section comprises Task 1.1 of the draft Fire Modeling Research Plan of Appendix A.

3.1 Models Reviewed

Of the variety of analysis tools that could be applied to the problem of modeling aircraft fires, the research, development, and application of fire field models were given highest priority based on the discussion in Section 2.3. Although other classes of fire models such as zone models have a role to play, fire field models represent the current direction of fire modeling research in the global fire community. Research was focused on several existing field models which could be applied to the aircraft fire problem: VULCAN (developed collaboratively by Sandia and SINTEF/NTH, and based upon the KAMELEON Fire model of Holen, et al., (1990)), the CFDS (Harwell) Flow3D model (Burns, et al. (1987)) which is now called CFX, the CHAM PHOENICS model, the CFDRC model from Computational Fluid Dynamics Research Corporation, and the University of Notre Dame Fire Model (referred to herein as NDFM) derived from the UNDSAFE code.

While the list of models reviewed is not exhaustive (i.e., there are a few other field models that could have been included), the above models formed a representative cross-section of available models. The models above cover a spectrum ranging from commercially available models (CHAM PHOENICS, CFDRC, CFDS Flow3D) to models developed and available through a university (NDFM). These models were chosen for review based primarily upon their capabilities relative to the classes of fire problems and capabilities of interest to FAA.

All of the above fire field models have a similar approach to the discretization and solution of the governing equations of heat, mass, and momentum transfer. They are all finite volume methods, and solve a discretized form of the governing equations on a grid. All of the models use similar turbulence submodels (k-ε type), although some have several possible turbulence submodels to choose from. The models solve similar equation sets, but there are some notable differences (as discussed below).

The CFDS and CFDRC models both have the capability of using non-Cartesian grid systems to define the fire domain and obstacles within it. The other models have been primarily developed and used in their Cartesian grid form. Non-Cartesian grid systems may be of some advantage for the simulation of aircraft fuselages in external pool fires, as they may result in improved determination of flow separation and hence pressure distribution around the vessel.

Some models have state-of-the-art combustion submodels (such as VULCAN's use of the Eddy Dissipation Concept (EDC), and CFDS' use of the EDC and PDF (probability density function) submodels). The NDFM model, on the other hand, does not have a combustion submodel in it, and relies on the user to input a heat release rate in the calculations.

The treatment of thermal radiation varied greatly among the models: from simple gas and surface radiation submodels to Monte Carlo algorithms, with a broad spectrum between the two.

3.2 Models Selected

Two models were selected for further evaluation, and application to problems including external pool fires as well as fires internal to the fuselage, since both types of fires are of interest for aircraft fire safety. The two models selected were the VULCAN fire field model and the CFDS Flow3D model.

VULCAN was selected because it had been the most widely applied to large-scale fires of any of the models reviewed. It was developed specifically to simulate fires, so very little adjustment of parameters or addition of submodels is necessary to directly apply it to aircraft fire problems of interest. Also, it has shown good agreement with experimental data for large pool fires.

CFDS Flow3D was selected as the best of the commercially available products. It has a wide range of features that make it attractive for fire modeling, although it hadn't been extensively applied to fire environments at the time it was reviewed. Of particular note are the various combustion and radiation submodels available in the model.

A license was obtained for CFDS Flow3D in order to apply it to problems of interest for aircraft fire research. However, several problems arose which made it difficult to get the model up and running on a local Silicon Graphics (SGI) Indigo workstation. It should be noted that every CFD code must be specifically configured for a particular computing platform. The problems encountered in getting CFDS Flow3D up and running are not unique to CFDS Flow3D, nor unique to SGI workstations, and could very well be encountered with any CFD code or computing platform. First, additional memory (RAM) was required to run the model (approximately 4 times that required to run VULCAN). Once the memory was purchased and installed, there were difficulties with the SGI FORTRAN compiler. This necessitated that an earlier version of FORTRAN be obtained from SGI in order to compile CFDS Flow3D. By that point in time, SGI had upgraded their FORTRAN compiler and no longer sold the earlier version. Unfortunately, no calculations were performed with the CFDS Flow3D software as part of this study due to time and resource limits.

As a result of this experience, VULCAN was used for all of the fire field model calculations performed as part of this study. No comparisons are directly made between VULCAN and any other fire field model, although some qualitative comparisons are made to the NDFM results for static pressure around a fuselage in Section 5. Comparisons are made between VULCAN and a risk-assessment-compatible fire model in Section 6.

Intentionally Left Blank

4.0 Sandia Fire Field Model Description

4.1 History

The fire field model VULCAN has been developed over the past 3 years by the Sandia Fire Science and Technology team at Sandia National Laboratories, Albuquerque, NM, in collaboration with the SINTEF Foundation and the Norwegian University of Science and Technology (NUST) in Trondheim, Norway. It is based on the KAMELEON Fire model developed by Professor B. Magnussen and colleagues at SINTEF/NUST during the past 2 decades. VULCAN is the name given to the Sandia version of the model. The collaboration between Sandia and SINTEF/NUST is ongoing at present, and VULCAN continues to be developed and applied as necessary.

Much of the validation data for VULCAN has been generated in the large-scale test series conducted by Sandia at NAWC (Naval Air Warfare Center, China Lake, CA) under the sponsorship of Defense Nuclear Agency (DNA). Since these test series have not yet concluded, validation of the model is an ongoing effort.

4.2 General Description

VULCAN uses an extension of the SIMPLE method of Patankar and Spalding (1972) to solve the conservation equations for mass, momentum, and energy transport on a finite difference grid. Second-order upwind differencing is used for the convective terms in the discretized partial differential equations. A staggered grid is employed to solve for the velocities in 3 dimensions. Both a k-ε model and a large eddy simulation model (LES) are available to model the turbulence in the flow.

The combustion model in VULCAN is based on Magnussen's Eddy Dissipation Concept (Magnussen (1989)). The Eddy Dissipation Concept (EDC) is a general concept for describing the interaction between the turbulence and the chemistry in flames. The EDC assumes that the combustion process occurs in the turbulent fine structures, which are modeled as perfectly stirred reactors. VULCAN presently assumes that the combustion process is irreversible and occurs infinitely fast, i.e., it does not include finite rate chemistry. However, an extinction test is included in the model. Local extinction is assumed to occur when the time scale for turbulent mixing (calculated by the model, but a function of the turbulent length scale, which is an input parameter) is less than the chemical time scale (input by the user). Incomplete combustion effects (due to the formation of CO and H_2) are accounted for.

The modeling of soot formation is based on a two-step process adapted by Magnussen (1981) from the work of Tesner, et al. (1971). The first step treats the formation of radical nuclei, and the second step the formation of soot particles from the radical nuclei. Once soot is formed, the EDC is capable of modeling the combustion of soot in the flame.

Thermal radiation of the combustion products (including soot) is modeled using the Discrete Transfer Method of Shah (1979). This method is used primarily because it represents an acceptable compromise between computational speed and accuracy for many problems. The soot and combustion gases are treated as a gray gas with an effective absorption and emission coefficient. The eddy viscosity near solid surfaces is calculated using the logarithmic wall function method. Convective and radiative heat transfer is modeled to objects in the flow field.

5.0 Calculations of Fire Impact on Fuselages

Calculations relevant to aircraft fire safety were performed with VULCAN to assess the ability of the model to simulate the important physical processes involved. These calculations included either a representative aircraft geometry or a large cylinder representative of an aircraft fuselage (sometimes referred to herein as a mock fuselage). Most of the work described herein is part of Task 1.2 in the draft Fire Modeling Research Plan of Appendix A. The exception is the work described in Section 5.3 below, which is part of Task 1.3. Other calculations using VULCAN for large pool fire simulations (without fuselages) have been documented elsewhere (e.g., Nicolette, et al. (1995), Gritzo, et al. (1995a)).

5.1 Simulation of Pool Fire Development Around an Aircraft

As an initial step in assessing the ability of VULCAN to simulate external pool fires involving aircraft, a demonstration calculation was performed. A three-dimensional (3D), transient calculation was performed of a mock aircraft exposed to an external jet fuel (JP4) pool fire in a cross-wind. A grid of 44 by 44 by 30 finite volume cells was used to represent a domain that was 160 m by 150 m by 80 m high. No penetration of the fire into the aircraft was permitted in the calculations. The intent of the simulation was to demonstrate whether VULCAN could calculate such a scenario, and to observe the resulting fire development from the simulation.

Ignition was specified to occur at 3 seconds in the calculations, which allowed for some evaporation of fuel from the pool. The pool evaporation rate was specified as 0.077 $kg/m/s^2$ based on typical experimental values for aviation fuel pools of this size (Gregory, et al. (1987)). The wind was specified at a constant velocity of 2.2 m/s (5 MPH), normal to the long axis of the fuselage. The standard k-ε turbulence submodel was used in the calculations, as the large-eddy simulation capability had not yet been incorporated into VULCAN.

All of the open boundaries of the calculation domain (except for the upwind boundary) were treated as constant pressure boundaries, which allows for inflow or outflow depending on the relative pressure difference across the boundary. The upwind boundary used a specified velocity. An atmospheric boundary layer profile was used to specify the distribution of velocity in the vertical direction along the upwind boundary:

$$V/V_{10} = (Z/Z_{10})^{0.118} \tag{1}$$

The velocity (V) at any vertical elevation (Z) is calculated based on the specified velocity at an elevation of 10 m (V_{10}) above the ground (2.2 m/s for this case).

Figure 1 illustrates the results. The figure depicts the time evolution of the fire around the aircraft in a cross-wind. The time evolution is clockwise in the figure, beginning in the upper left hand frame. The view in the figure is looking at the nose of the plane, and the wind is from the left. The initial fuel spill (approximately 100 m^2) is placed under the wing of the aircraft as shown. Although Figure 1 is a 2D slice through the calculation domain, the maximum temperature along the line of sight is projected to the plane of view. This yields a reasonable picture of the 3D fire development.

The first frame of Figure 1 (upper left hand corner) illustrates the temperature field right at the point of ignition (time = 3 seconds) before there has been any significant fire development. By the 6 second mark (second frame, proceeding clockwise), the evaporated fuel above the pool has been ignited and the flames engulf the bottom and sides of the fuselage. The effect of the cross-wind is clearly seen in the second and third frames to tilt the flames toward the leeward direction, and causes the flames to spread over a significantly larger ground area (almost twice as large) than the pool occupies. At the 15 second mark (last frame) it can be noticed that the fire has now set up its convective plume, and although the wind does affect the flame shape somewhat, the fire plume has gained sufficient strength to reduce the angle at which the flames are tilted. It is also interesting to note that a portion of the flames remain attached to the underside of the fuselage, illustrating the tendency of objects to act as flame-holders (even large objects such as fuselages). Such local effects, due to the interaction of fire, fuselage, and wind, are critical to the prediction of important phenomena of interest to aircraft safety, such as fuselage burn-through.

Calculations were performed with several other domain sizes to investigate the effect of placing the calculation boundaries nearer and farther away from the aircraft. For domains that were significantly smaller than the one reported above, non-physical flows resulted. The fire plume began to bend into the wind, as the calculations began to draw free air more easily from the side boundaries. This result can be understood in the following manner. When a wind is present, a velocity is specified along the upstream boundary. This fixes the upstream velocity at the boundary. For high winds, this does not present any difficulty, or for calculations with a large domain. However, for low wind speeds and small calculation domains, the air which the fire naturally entrains from the upwind direction can exceed the low wind speed specified along the boundary. Under these conditions, the fire wants to draw more free air from the upstream boundary than the specified velocity boundary condition will permit. Thus the upstream boundary condition becomes physically unrealistic. In order to overcome this lack of air in the upwind direction, the calculated fire begins to pull air from whatever direction it can, leading to non-physical results. When the domain is made large enough (such as the one reported above, and larger) such effects do not occur. The calculated fire is able to entrain sufficient air in the upwind direction, and

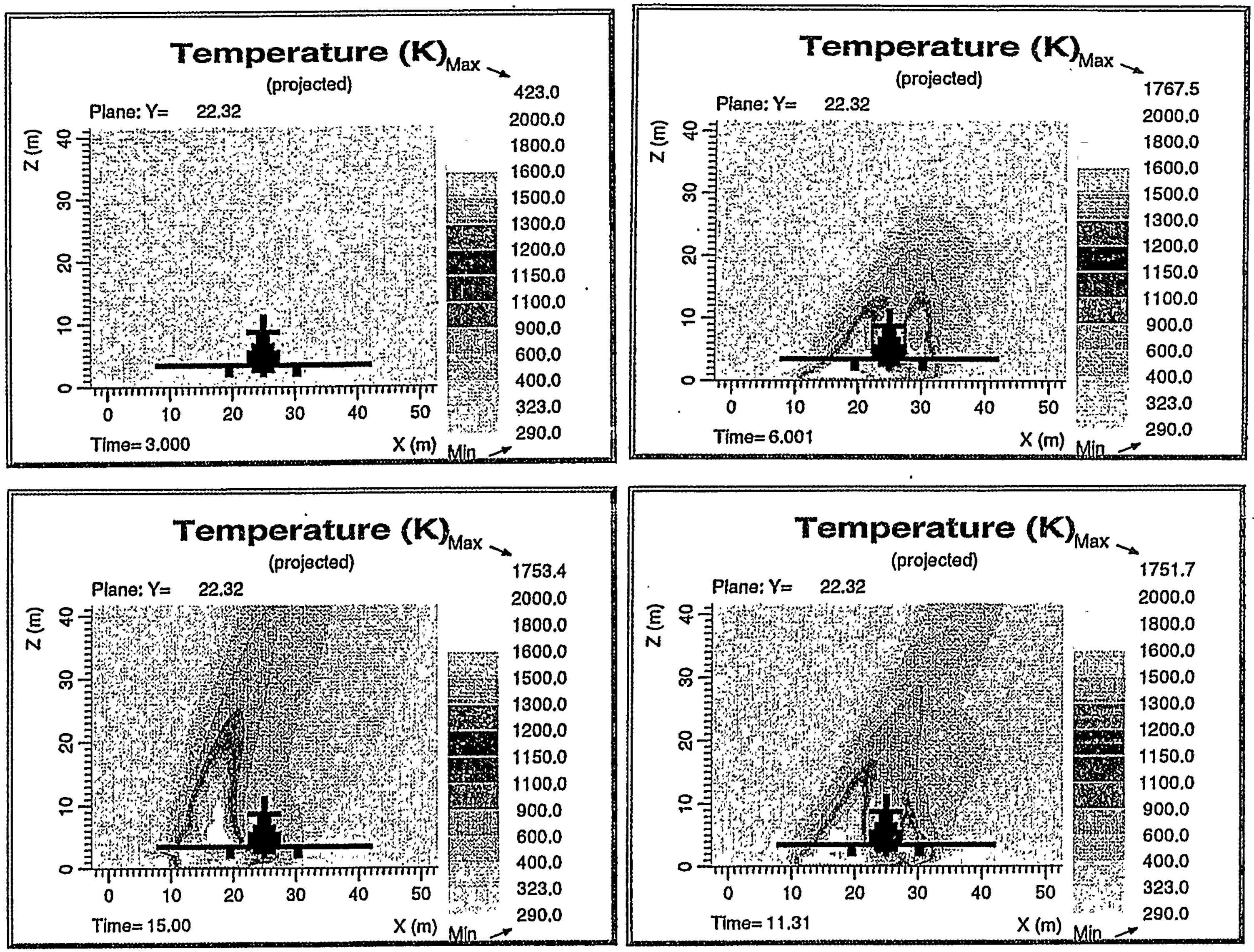


Figure 1. Simulated Time-Evolution of Fire Around an Aircraft in a 2.2 m/s Cross-wind. Sequence is clockwise, from upper left.

non-physical results are avoided. It should be noted that the amount of domain that is required to avoid non-physical results will be a function of the wind speed, the fire size, and the manner in which the boundary conditions are implemented in the fire model.

The above results have been animated and transferred onto VHS videotape to better illustrate the time evolution of the fire. These results illustrate the ability of field models such as VULCAN to simulate the important physical processes that are involved in aircraft fires, and the insights that can be gained through the application of fire field models to such problems.

5.2 Simulation of NAWC Test with Mock Fuselage

A simulation using VULCAN was conducted of a test performed for DNA at NAWC involving a mock fuselage in a pool fire with wind. The simulation results were compared to the test data, and are presented fully in Gritzo, et al. (1995b). Since the results have already been presented elsewhere, they are only briefly summarized in this section.

The mock fuselage used in the test was a 3.66 m diameter steel culvert, 20 m in length (referred to hereafter as the fuselage). The fuselage was between 0.23 m and 0.6 m above the fuel pool (the elevation changed as a function of sagging of the culvert and height of the fuel). It was positioned at the leeward edge of the pool, with the wind direction 23 degrees away from the normal to the long axis of the cylinder. For a portion of the test, a relatively constant wind speed of 10.2 m/s (~22 MPH) was measured. The fuel was an 18.9 m diameter pool of JP8.

Simulations were conducted using a 42 X 40 X 30 grid to represent a physical domain of 150 m X 150 m X 60 m (high), with grid points clustered near the fuselage and the fuel surface. It was necessary to cluster the grid points near the fuel surface in order to resolve the small gap (0.23 m) between the bottom of the fuselage and the top of the fuel. Evaporation from the pool was calculated by VULCAN as a function of the absorbed thermal radiation from the flames. The standard k-ε turbulence submodel was used in the calculations, as the large-eddy simulation capability had not yet been incorporated into VULCAN.

As in the previous calculation, the open boundaries were treated as constant pressure boundaries, with the exception of the upwind boundaries. Since the wind was at an angle to the fuselage, which was aligned with the cartesian grid, velocity components had to be specified along both the x and y upwind boundaries. As given in Equation 1, an atmospheric boundary layer profile was assumed for both the x and the y velocities along the boundaries.

Qualitatively similar flame coverage was predicted by the model relative to that observed during the test (and recorded on videotape). The largest area of flame coverage was on the leeward side of the fuselage (Figure 2). Very high heat fluxes (250-300 kW/m^2) were predicted by the model on the leeward side of the fuselage. This was in good agreement

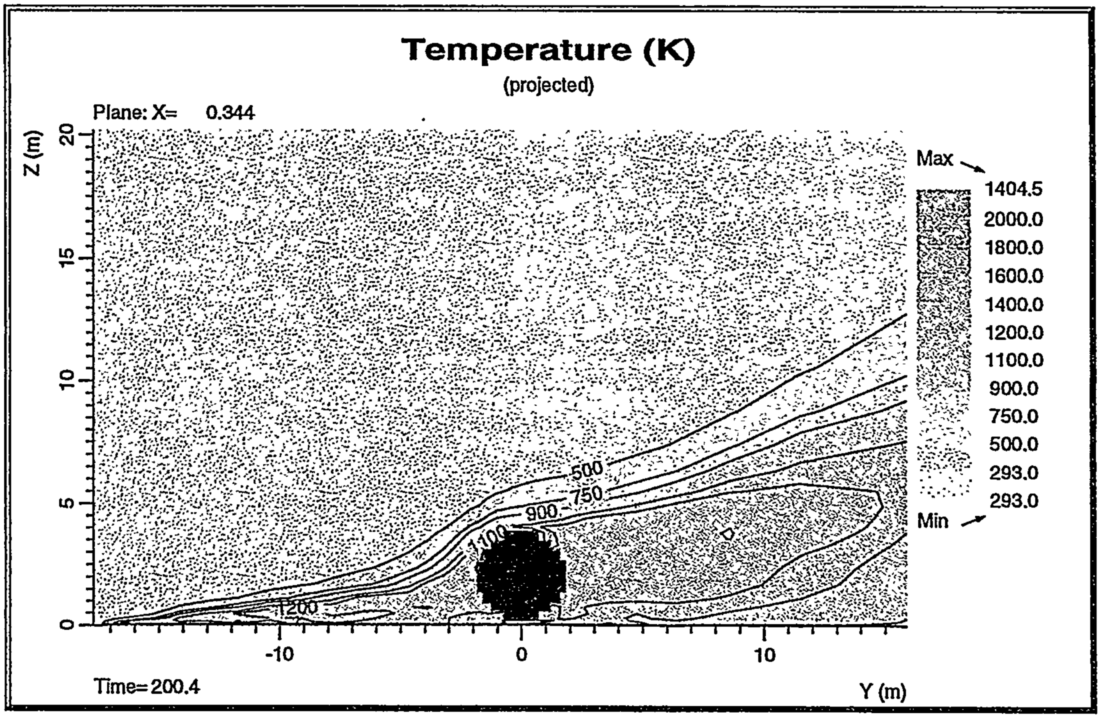


Figure 2. Calculated Flame Coverage Around Mock Fuselage in NAWC Test With 10.2 m/s Wind. Fuselage on Leeward Side of Pool. Pool extends from Y=-17.2 m to Y=+1.7 m.

with the measured heat fluxes on the leeward side, which were in the same range. It was also observed in the tests that inconel thermocouples on the leeward side of the fuselage melted (the melting temperature for inconel is approximately 1665 K). This further attests to the high heat fluxes on the leeward side. The simulation results show the formation of a large, stable vortical structure on the leeward side of the fuselage, in agreement with video recordings of the fire. This structure produces intense local turbulence, and consequently results in a well-mixed region of fuel and air. As a result, a region of intense combustion is predicted on the leeward side of the fuselage, with very high temperatures and heat fluxes.

The model does a reasonable job of predicting the qualitative features of the radiative heat flux distribution to the fuel pool surface. Quantitatively, the calculated heat fluxes to the fuel pool tend to be somewhat lower than measured. The interested reader is referred to Gritzo, et al. (1995b) for more details.

Since there was some sagging of the fuselage during the test (as determined by before and after measurements), the calculations were conducted for two different fuselage heights above the pool: 0.23 m and 0.6 m. The results of the simulations were sensitive to this value. The higher elevation resulted in less flame coverage over the top of the fuselage, and more of the fuel blown under the fuselage and burned on the leeward side. The lower value of elevation was selected for the comparisons, since this value was representative of the fuselage elevation measured at the end of the fire, and was thought to best represent the fuselage height during most of the test.

These results demonstrate the importance of the coupling between the fuselage and wind in a fire. Relatively small changes in fuselage shape during a fire can have a significant effect on the flame coverage, and subsequently influence the local heat fluxes to the fuselage. This illustrates the need to eventually couple the structural response of the fuselage to the fire model. In this regard, it should be noted that in the test and simulations the mock fuselage remained intact, but that a typical aluminum fuselage would have melted very quickly on the leeward side.

It is also interesting to note that large vortical structures are formed in large pool fires in a cross-wind even when there is no object in the fire. These large vortical structures have a tremendous effect on the local heat flux distribution to an object (such as a fuselage) in a fire. Consequently, a spatially averaged heat flux does not adequately represent the local heat flux distribution within a large pool fire in a cross-wind. Large spatial variations in heat flux exist within the flame zone of large pool fires, both in the test data and in the numerical results. The interaction of the fire, wind and fuselage determines which structures form, where they form, and which one dominates the local heat flux distribution.

5.3 Large-Eddy Simulation of Pool Fire Development Around a Mock Fuselage

In the calculations presented thus far, the standard k-ε turbulence submodel was used in VULCAN. As demonstrated by the previous results (and those of Tieszen, et al. (1996)), this model produces very reasonable results for large external pool fires with (and without) large objects such as a fuselage present. However, there are limitations involved with the use of any model, and the k-ε turbulence submodel is no exception. In order to investigate the effect of relaxing the inherent limitations which the k-ε turbulence submodel introduces into VULCAN's results, a study was performed in which the effect of turning off the k-ε turbulence submodel in VULCAN, and incorporating a large-eddy simulation (LES) submodel for turbulence, was investigated. The results of an LES calculation involving a mock fuselage in a fire are presented in this section. This work (and that of Tieszen, et al. (1996)) is part of Task 1.3 of the draft Fire Modeling Research Plan of Appendix A.

A detailed discussion of the differences between the k-ε turbulence submodel and an LES submodel (see for example, Tieszen, et al. (1996)) is beyond the scope of this report. For the purposes herein, it suffices to mention the following differences between the two submodels. The k-ε turbulence submodel averages all of the turbulence fluctuations (including all turbulence length scales) over time. In contrast, the LES submodel does not use a temporal averaging of the turbulence, but rather a spatial averaging for all turbulence length scales that are smaller than the grid size used in the calculation. Since the LES submodel does not time-average the turbulence, it can be used to simulate time-varying turbulent structures (that are larger than the grid size used in the calculations). The ability to simulate time-dependent turbulent structures (such as the puffing of a large pool fire) could be important for the prediction of such phenomena as remote ignition, flame spread, and pressure distribution around a fuselage in a fire.

Once that a large-eddy simulation capability had been incorporated into VULCAN, scoping calculations were performed to assess the abilities of the model. In Tieszen, et al. (1996), the results of calculations to predict the puffing frequencies of pool fires of various diameters are compared with experimental data from Cetegen and Ahmed (1993). A figure from that work is reproduced here (Figure 3). The results indicate very good agreement

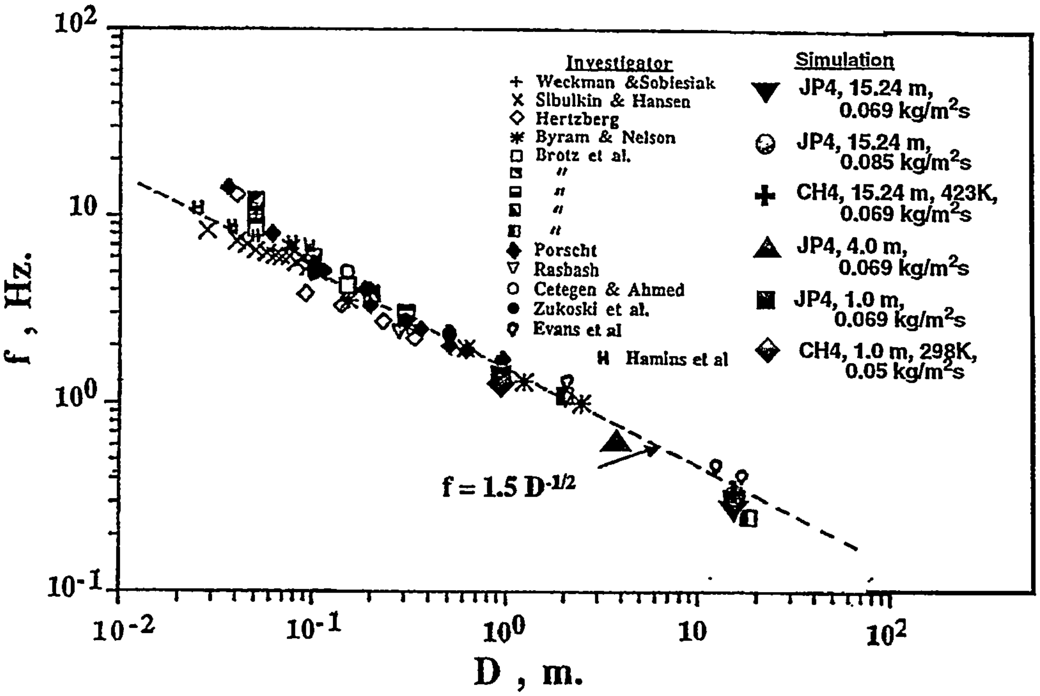


Figure 3. LES Prediction of Pool Fire Puffing Frequency versus Pool Diameter: Comparison to Data. Taken from Tieszen, et al. (1996).

between the calculations and data, and demonstrate the usefulness of the LES submodel.

To illustrate the abilities of the VULCAN LES submodel for fires involving aircraft, a simulation was performed with the LES submodel for a pool fire with a mock fuselage in it. The input parameters for the problem were identical to those discussed in the previous section, with the following exceptions. The fuselage was assumed to be at the high elevation (0.61 m) above the ground. The wind speed was 1.4 m/s (3 MPH), and the wind direction was assumed to be normal to the longitudinal axis of the fuselage. Finally, the LES submodel discussed in Tieszen, et al. (1996), was used to model the large-scale turbulent structures, instead of the k-ε turbulence submodel.

Due to the time-varying nature of the LES submodel, no steady flame shape was obtained in the calculations. Large puffs of flame and smoke periodically formed and rolled up in the calculations. The fuselage was periodically immersed in flames and then partially uncovered. Figure 4 shows the flame shape at 4 minutes following ignition. It can be seen in this figure that a large puff has just lifted off of the ground plane. Flames appear on the leeward side of the fuselage. A separation region appears near the top, leeward side of the fuselage, in which there are no flames present. A horizontal slice through the fuselage at an

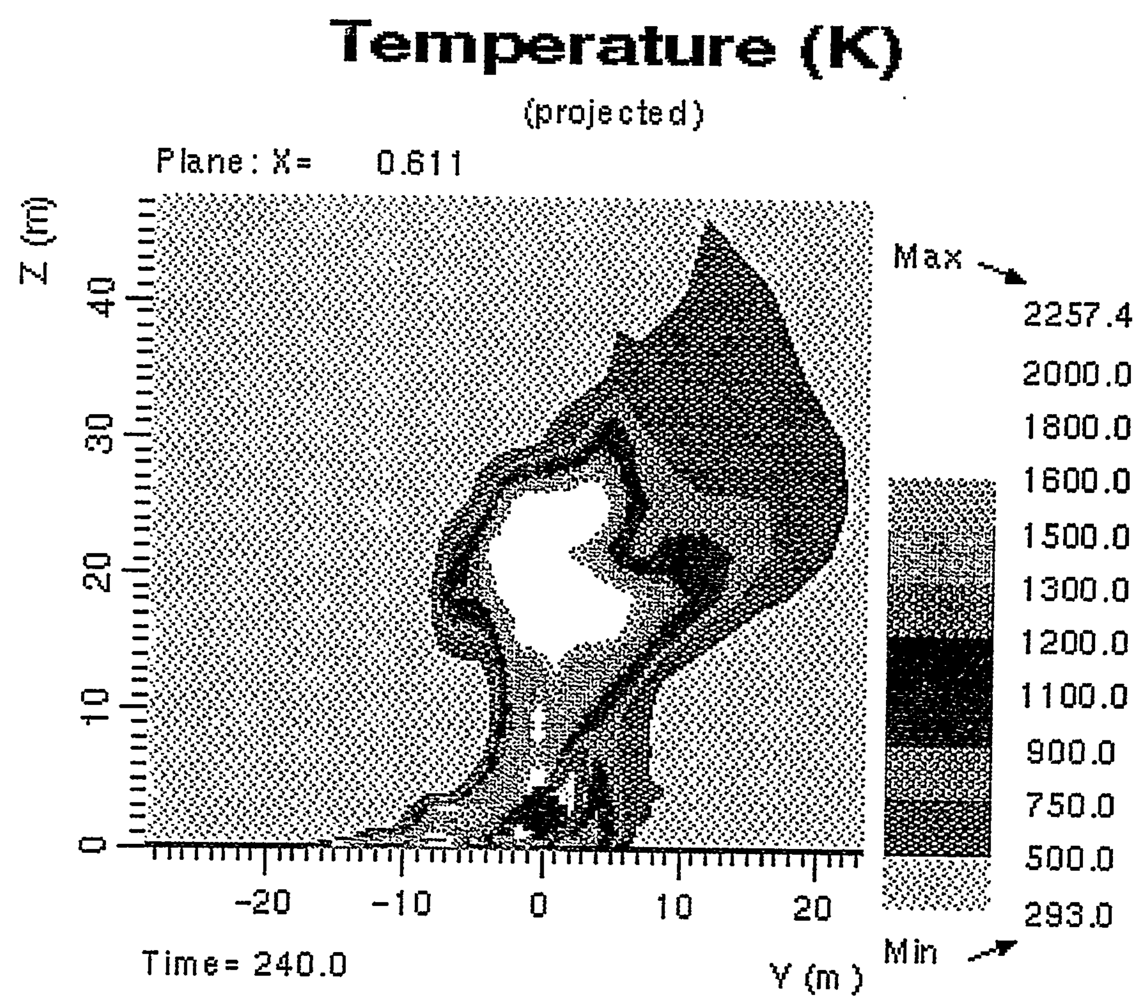


Figure 4. LES Calculation of Flame Shape Around Mock Fuselage in 1.4 m/s Cross-wind: Time = 240 seconds.

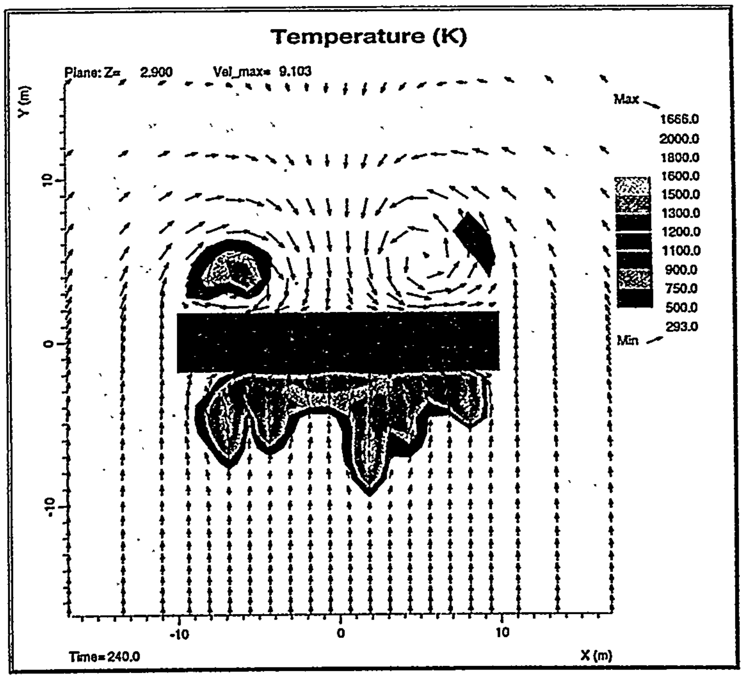


Figure 5. LES Calculation: Horizontal Slice Through Fuselage at Time = 240 seconds Showing Vortices on Leeward Side of Fuselage

elevation of 2.9 m above the ground is presented in Figure 5. Velocity vectors in this figure indicate that the flames that appear on the leeward side of the fuselage are associated with two columnar counter-rotating vortices. These vortices have been observed in large-scale tests with similar wind speeds. In the LES calculations, these vortices formed periodically on the leeward side of the fuselage, in agreement with observations made during low wind speed tests.

At other instances in time, the fuselage did not have flame coverage along the leeward side, as illustrated in Figure 6 at a time of 7 minutes after ignition. As a result of the large fluctuations in flame shape evident in the preceding figures, the heat fluxes to the fuselage also varied greatly with time. Additionally, large spatial variations in heat flux to the fuselage were also observed in the calculations as a result of the large variations in flame shape and coverage over the fuselage. The radiative heat fluxes to the fuselage at a time of 7 minutes after ignition are shown in Figure 7 corresponding to the flame shape shown in the previous figure. It should be noted that the radiative heat fluxes in large, sooty pool fires are generally much greater than convective heat fluxes, especially for objects as large as aircraft fuselages.

As seen in Figure 7, very high heat fluxes are predicted on the upwind side of the fuselage as a result of the large region of flame coverage, with the highest heat flux roughly 250 kW/m^2. High heat fluxes are predicted along the upwind portion of the top side, with a peak of 175 kW/m^2. The heat fluxes to the leeward portion of the top side are substantially lower,

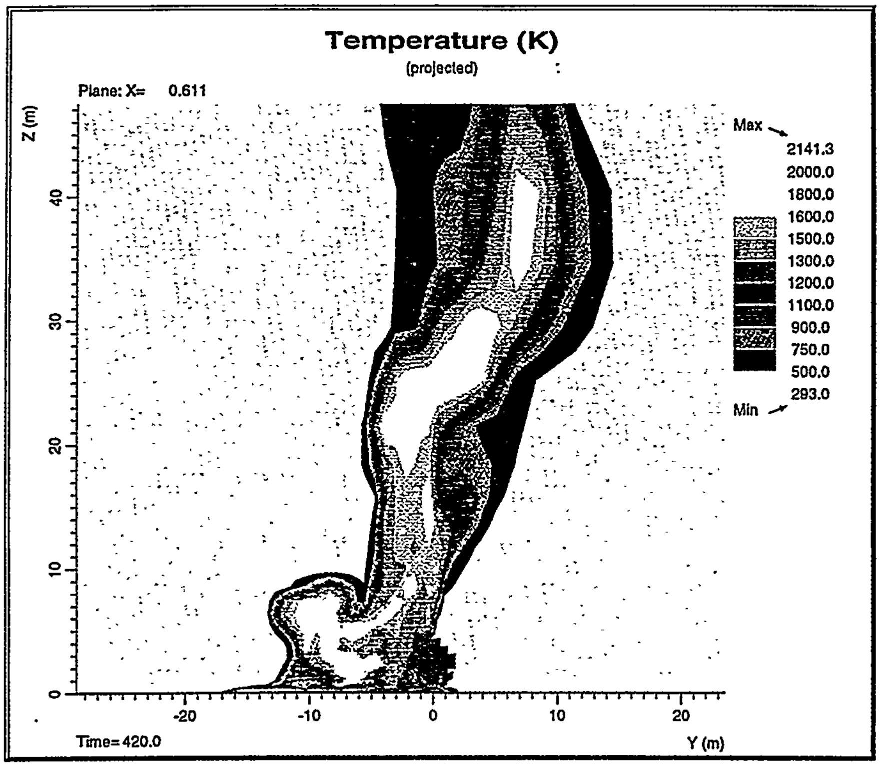


Figure 6. LES Calculation of Flame Shape Around Mock Fuselage in 1.4 m/s Cross-wind: Time = 420 seconds.

due to the lack of flame coverage on the leeward side. Heat fluxes to the leeward side of the fuselage are very low (< 16 kW/m^2), and result primarily from a small amount of flame coverage near the bottom of the fuselage on the leeward side. The bottom of the fuselage has heat fluxes that are generally less than 40 kW/m^2, although a hot spot is seen of 148 kW/m^2 at this instant in time. The calculated convective heat fluxes to the fuselage were generally less than 10 kW/m^2.

These calculations demonstrate the large temporal and spatial variations in flame shape and heat fluxes that can occur when fuselages are involved in fires. Developing an understanding of the dynamic interactions of a fuselage, fire, and wind is a complex task. This complexity arises from the coupling of the nonlinear physical phenomena, primarily: momentum transport, combustion, soot generation, and radiative heat transfer.

5.4 Calculations of Pressure Field Around a Fuselage in Wind

A limited number of calculations of the static pressure field and velocity field around a fuselage in a cross-wind (without fire) were performed. These will be discussed herein, following the presentation of some background information. In all cases, the fuselage (or

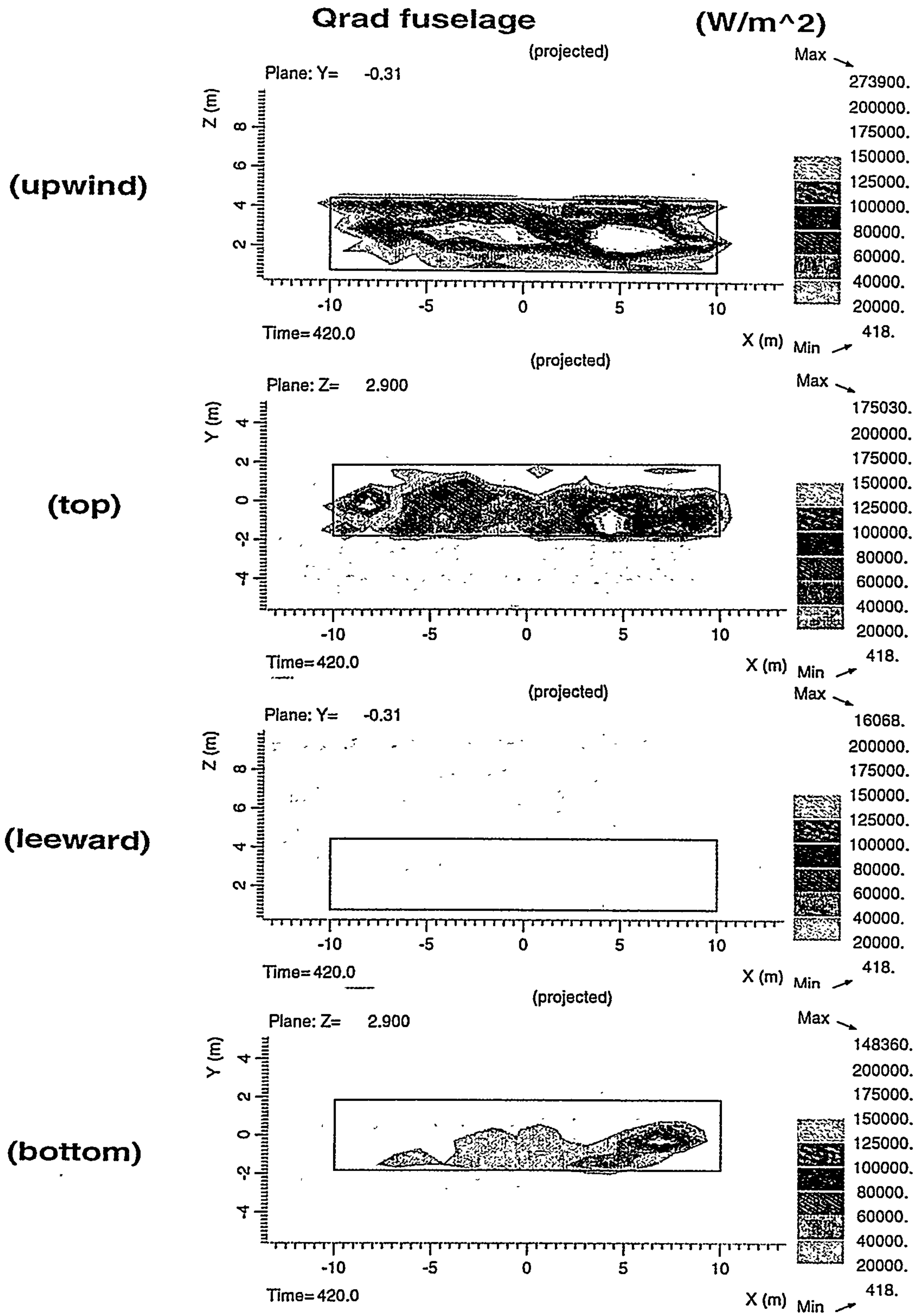


Figure 7. LES Calculation of Radiative Heat Fluxes to Mock Fuselage in 1.4 m/s Cross-wind: Time = 420 seconds.

cylinder) referred to is located a small distance above or right on the ground. The phrase "on the ground" is used to refer to an aircraft that is not in flight.

The static pressure distribution around an aircraft fuselage on the ground plays a central role in determining the response of an aircraft and its passengers in an accident involving fire. The pressure field around the fuselage is determined by the interaction of wind, fire, fuselage, and the ground plane. For an external pool fire, this interaction greatly influences the location of turbulent mixing regions which result in local hot spots (see the discussion in the preceding section). The static pressure field around the fuselage also serves as a boundary condition for flows through the hull and any penetrations. This could be of significance in two ways. First, fuselage burn-through times can be significantly influenced by whether the surrounding pressure field pulls the fire in toward the fuselage (and its penetrations) or pushes the fire out away from the fuselage. Fuselage burn-through time is an important factor in passenger escape time from aircraft exposed to an external pool fire. Such considerations could enable a pilot to orient the aircraft most effectively relative to the wind direction and fire location to maximize the fuselage burn-through time. Second, in the evacuation of an aircraft exposed to an external pool fire, if doors (away from the fire) are opened on the pressurized side of the fuselage, smoke ingression into the cabin could occur more slowly than if doors are opened on the opposite side of the fuselage.

There have been several previous computational investigations of the flow field and pressure field around a fuselage involved in a fire on the ground. They will be briefly summarized here to provide background information. Kou, et al. (1985), performed a two-dimensional simulation and observed oscillations in the pressure field when buoyancy effects are strong. The oscillations did not appear when strong winds were present (strong relative to the buoyancy of the fire). The fire source for the calculations was a buoyant heat source distributed over a predetermined volume in space. The effect of this assumption on their results is unknown. Galea and Hoffman (1995), performed a computation of the effect of an internal fire on a Boeing 737 in a cross-wind. The flow in the internal cabin and the flow over the exterior of the hull were solved, using a specified wind speed at the boundary of the computational domain. Velocity and temperature fields (internal to the cabin) are presented for the scenarios of: wind only, fire only, and both fire and wind. It is interesting to note that the computational domain extended for less than one half of the fuselage diameter on either side of the fuselage, and less than one quarter of the fuselage diameter over the top of the hull. The results presented in Section 5.1 above and here below indicate that significantly more spatial domain is required in the computations to accurately capture the pressure (and hence velocity) field around a fuselage in cross-flow.

The accurate calculation of the static pressure field around a fuselage in a cross-wind is very difficult. There are several factors that combine to make this computation very challenging. First, for a typical fuselage diameter of 3.5 m, and winds of 1-20 m/s (2-52 MPH), the Reynolds number (based on diameter) of the fuselage, Re, is $1x10^5$ - $5x10^6$. This range of Reynolds number (Re) is, coincidentally, the range of Reynolds number over which transition occurs in the boundary layer from laminar to turbulent flow for a smooth cylinder in cross-flow (see Gerhart and Gross (1995), pp. 522-529). The extent to which present day turbulence models can capture transitional flow is subject to question, as they were generally developed for fully turbulent flow (and at best, a re-laminarization of turbulent flow). For example, the constants in the well-known k-ε turbulence model are based on fully developed turbulence which is isotropic. One cannot expect that such a model will be highly accurate when applied to a transitional flow.

Secondly, form drag (or pressure drag) generally dominates flows with Reynolds numbers in excess of 10^3, thus minimizing the impact of the surface drag. However, this is not true in the transitional region (Gerhart and Gross (1995), p. 522, Schlichting (1979), p. 664), and surface roughness becomes important even at high Reynolds numbers (10^5 - 5×10^6). This sensitivity to surface roughness is indicative of transitional flows in general to minor disturbances. Transitional flows are also known to be highly sensitive to slight fluctuations in boundary conditions, such as might arise from fluctuations in the wind speed or from vortex shedding from an upstream wing or engine surface. This sensitivity of transitional flows to minor disturbances is also important for the present study from the standpoint that a Cartesian grid is used for the calculations presented herein. The use of a Cartesian grid may introduce disturbances into the calculations, due to the sharp corners of the grid cells, and affect the location of the flow separation.

Thirdly, relatively low values of differential pressure drive the flow field resulting from a fire in a cross-wind. For example, a wind speed of 0.447 m/s (1 MPH) yields a stagnation pressure of 0.05 Pa. A wind speed of 5 m/s (13 MPH) yields a stagnation pressure of 15 Pa. For comparison, the hydrostatic pressure difference across the fuselage from bottom to top is about 40 Pa, and atmospheric pressure is 101,300 Pa at sea level. Measurement of such low pressures is very difficult, given the extreme conditions of a fire environment. This makes it difficult to quantify the errors associated with computation of the static pressure field around a fuselage.

In spite of these known difficulties with computational prediction of pressure field around a fuselage with wind and fire, an attempt was made to estimate the pressure field for a few scenarios using VULCAN with the k-ε turbulence model. Initial calculations were only for a cross-wind over a fuselage, without a fire. The eventual goal of such calculations is to be able to include both the fuselage and the external pool fire in the calculations. While Galea and Hoffman (1995) have focussed on internal fires, the longer term goal of this work is to include all of the fire domain of interest: the fuselage plus the external pool fire, plus additional domain to ensure boundary conditions have been properly applied (see the discussion in Section 5.1 regarding the influence of boundary conditions on the fire development over a fuselage). While reaching this goal will require more advanced computing architectures than are currently available, such architectures are currently under development (e.g., the Department of Energy's Accelerated Strategic Computing Initiative). It was originally planned that the predictions herein would be compared to actual field measurements of the pressure field around fuselages in cross-winds, with and without fire. However, two different test efforts have had difficulties in obtaining accurate full-scale pressure data, and no comparisons to experimental data can be made to date.

In view of the difficulties associated with the computation of static pressure fields around fuselages in cross-winds with fire, several preliminary calculations were made with simpler scenarios to serve as a basis for comparison. All calculations (without fire) were carried out until a steady state had been reached (defined as less than a 2% fluctuation in stagnation pressure). The k-ε turbulence model was used in all of the calculations. A preliminary calculation (2D) was made of a cylinder 3.5 m in diameter, in a cross-wind. Second-order upwind differencing was used for the advective terms in the momentum equations. No ground plane was used, and the cross-flow was assumed to have a uniform velocity distribution of 3.1 m/s (7 MPH), corresponding to Re=8×10^5 (transitional flow). The ground plane was not included in the computations in order to investigate whether the pressure distribution would be symmetrical (as anticipated), and to provide a frame of reference for the ensuing calculations. There is no hydrostatic pressure in these calculations, so that the pressures presented are the static pressures (Pa).

The computed pressure distribution for this case is shown in Figure 8. The wind is from left to right in the figure, and the front edge (at stagnation) is seen to have the maximum pressure. Continuing around the cylinder, the pressure becomes negative just before reaching +/- 45 degrees (zero degrees being stagnation). The minimum pressures occur at

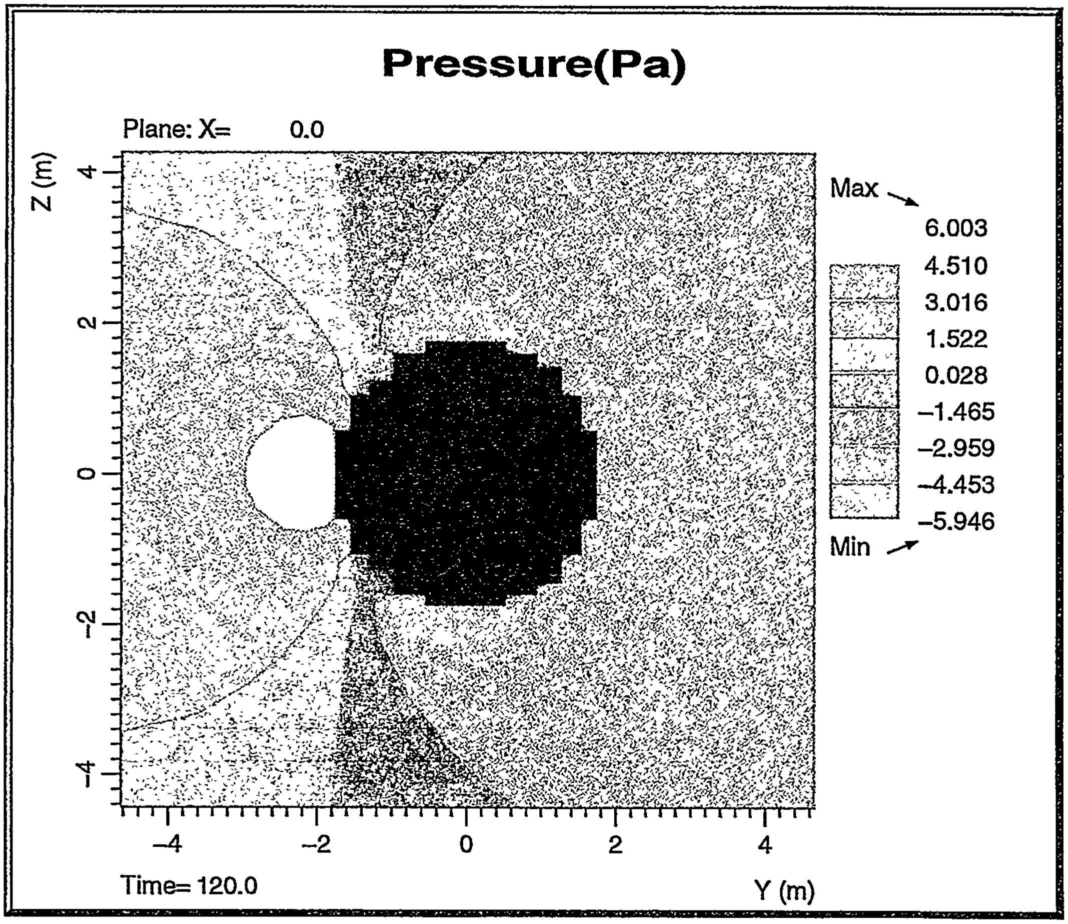


Figure 8. Static Pressure Distribution Around Cylinder in Cross-Flow: Re=8×10^5. Minimum Pressure Occurs at +/- 70 degrees from Stagnation.

roughly +/- 70 degrees, and the pressure remains negative through +/-180 degrees. The results also exhibit a fairly symmetrical profile as anticipated. The stagnation pressure is within 4% of the expected value for an inviscid flow. Figure 9 shows the velocity field near

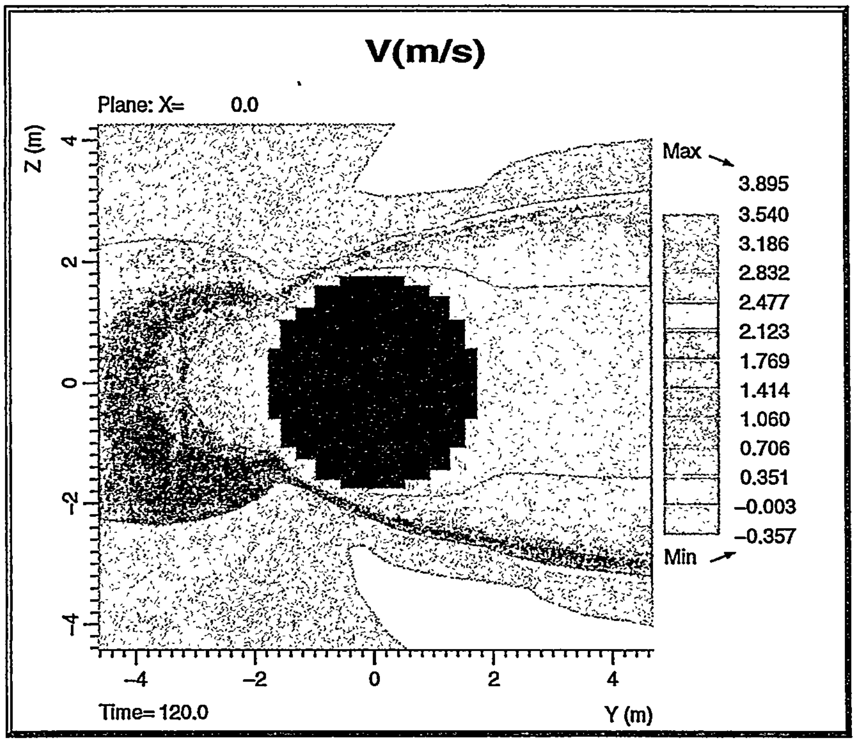


Figure 9. Velocity Distribution Around Cylinder in Cross-Flow: Re=8x10^5. Separation Occurs near +/- 77 degrees.

the cylinder, and indicates that separation occurs near +/- 77 degrees.

Schlichting (1979, pp. 172-3) reports that laminar boundary layers separate from smooth cylinders at +/- 82 degrees, and turbulent boundary layers at roughly +/- 109 degrees. Also, the location of minimum pressure is reported to be near +/- 70 degrees for a laminar boundary layer, and +/- 90 degrees for a turbulent boundary layer. The VULCAN results are in surprisingly good agreement (given the previous discussion) with those reported by Schlichting for cylinders with laminar boundary layers. There is also good agreement with the expected pressure distribution (Schlichting (1979), Figure 1.10) for a laminar boundary layer over the forward half of the cylinder. However, over the latter half the calculated pressure distribution agrees with that reported for a turbulent boundary layer, indicating that boundary layer transition has occurred along the cylinder surface.

A second calculation was performed for the same cylinder, but now using a ground plane 2 m below the cylinder. The results (Figure 10) indicate a slightly higher value of the stagnation pressure (12%), and lower values of the minimum pressures (21%). In addition, there is a slight skewing of the pressure profile resulting from the ground plane, and the area of minimum pressure extends farther downstream underneath the cylinder. This asymmetry in the pressure profile is in qualitative agreement with the results of Kou, et al. (1985). The velocity field is shown in Figure 11, and velocities underneath the cylinder are

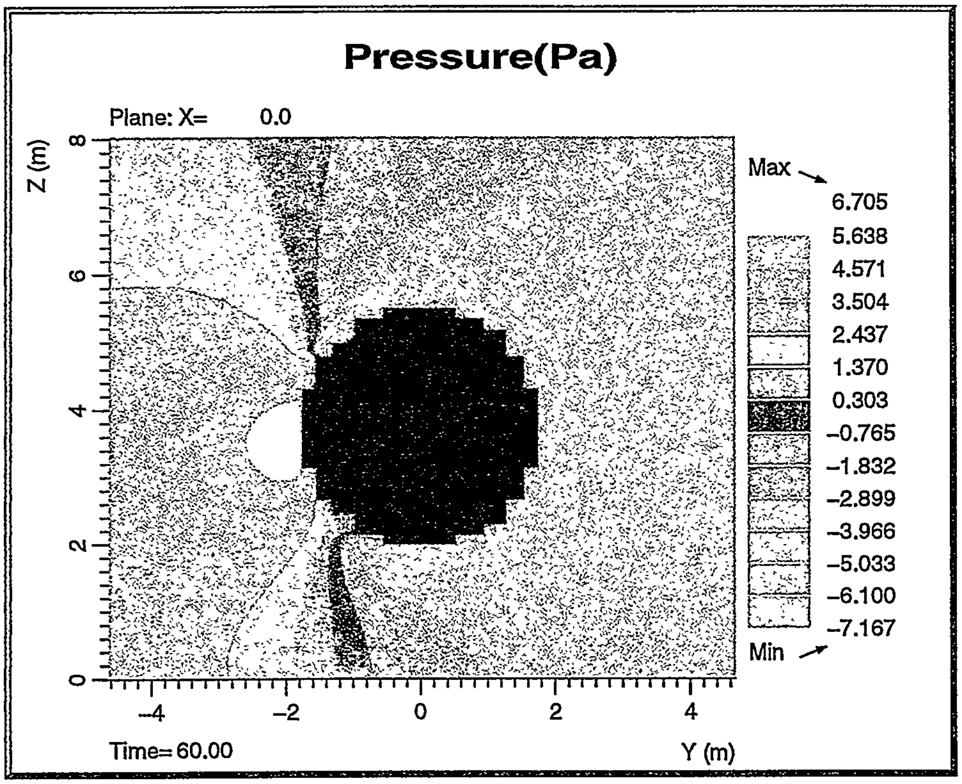


Figure 10. Static Pressure Distribution Around Cylinder in Cross-Flow, With a Ground Plane: Re=$8x10^5$

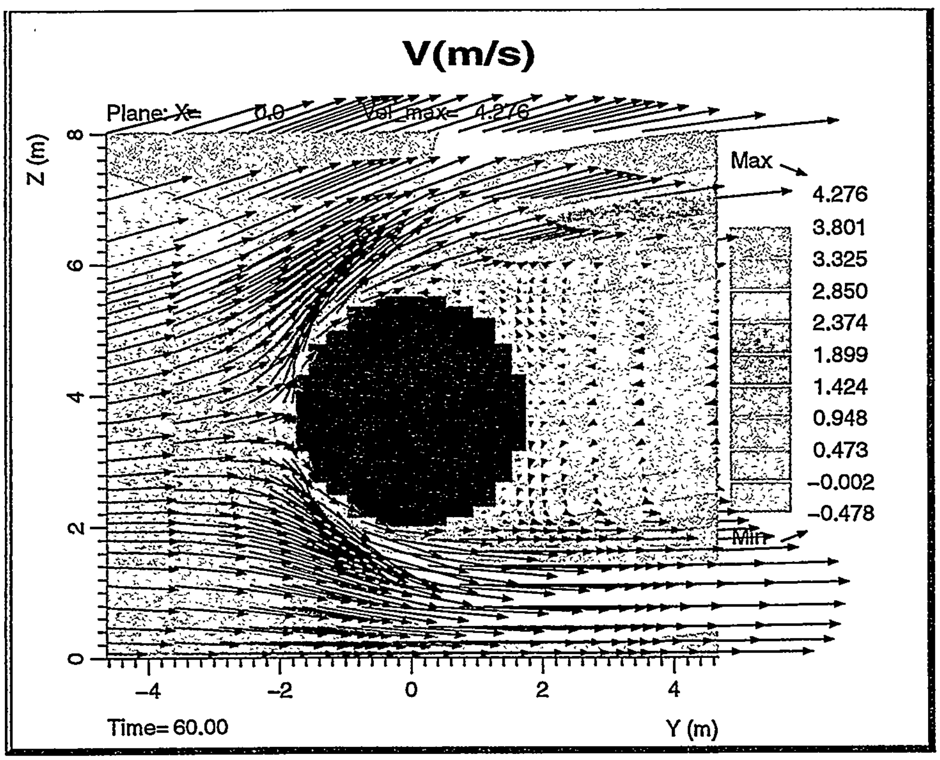


Figure 11. Velocity Field Around Cylinder in Cross-Flow, With Ground Plane: Re=$8x10^5$. Separation underneath is delayed beyond 90 degrees.

higher by 10% compared to the previous case. Over the top of the cylinder, separation occurs before +90 degrees, as in the case without a ground plane. But underneath the cylinder, separation is delayed beyond -90 degrees, to approximately -105 degrees. This value is indicative of a turbulent boundary layer, perhaps triggered by the increased flow velocity beneath the cylinder, and illustrates the sensitive nature of transitional flows.

Another calculation with a ground plane was conducted using a more refined grid (approximately twice as fine in the vicinity of the cylinder). The results for stagnation pressure are 2% lower than for the coarser grid case, while the minimum pressure is 18% lower. Since the locations of separation and minimum pressure were identical to the coarser grid solution, the coarser grid was used in the remaining calculations.

To investigate the effect of a distribution in wind velocity (as would be expected in a typical atmospheric boundary layer), a vertical profile was given to the wind speed. The wind speed was assumed to vary as the ratio of elevation relative to a standard elevation of 10 m, as given previously in Equation 1. A wind speed of 3.1 m/s (7MPH) was specified at the 10 m elevation. The results are shown in Figure 12. The lower wind speeds near the ground

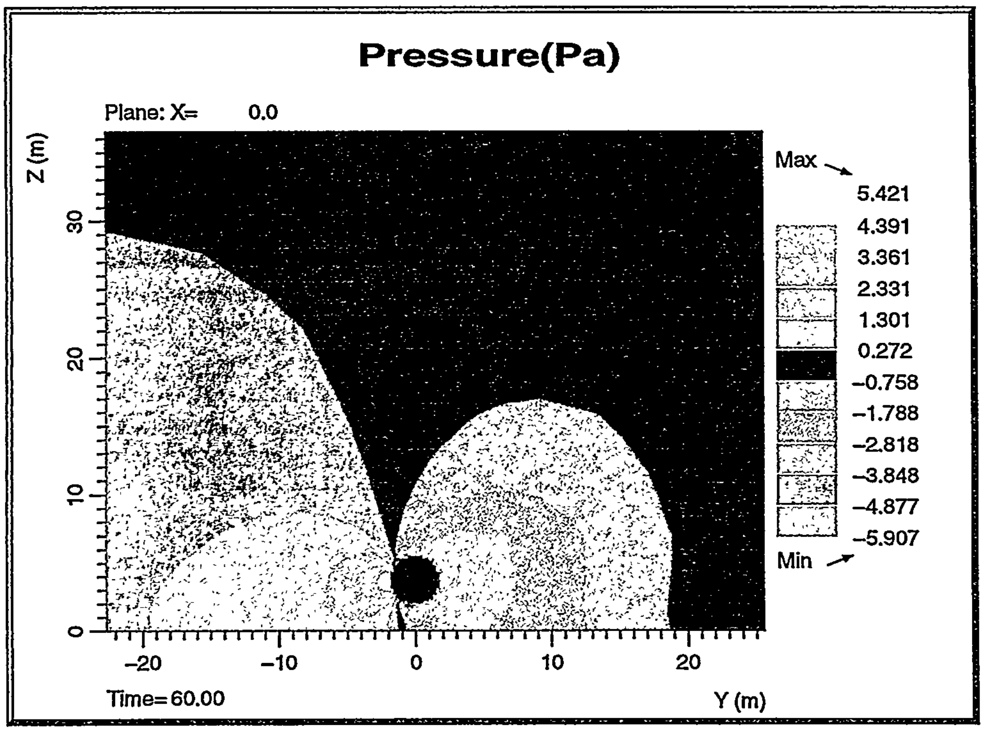


Figure 12. Static Pressure Distribution Around Cylinder in Cross-Flow, With Ground Plane and Atmospheric Wind Profile: Re=8×10^5

(below 10 m elevation) result in a lower stagnation pressure on the fuselage (20% lower). The minimum pressure around the fuselage decreases by 18% from that of the uniform profile case (Figure 10). A plot of the velocity contours associated with this calculation are shown in Figure 13. It is interesting to note the area of the flow which the fuselage impacts. As expected, there is a large wake region downstream of the cylinder (more than 3 diameters before negative velocities are no longer predicted). It is somewhat surprising to

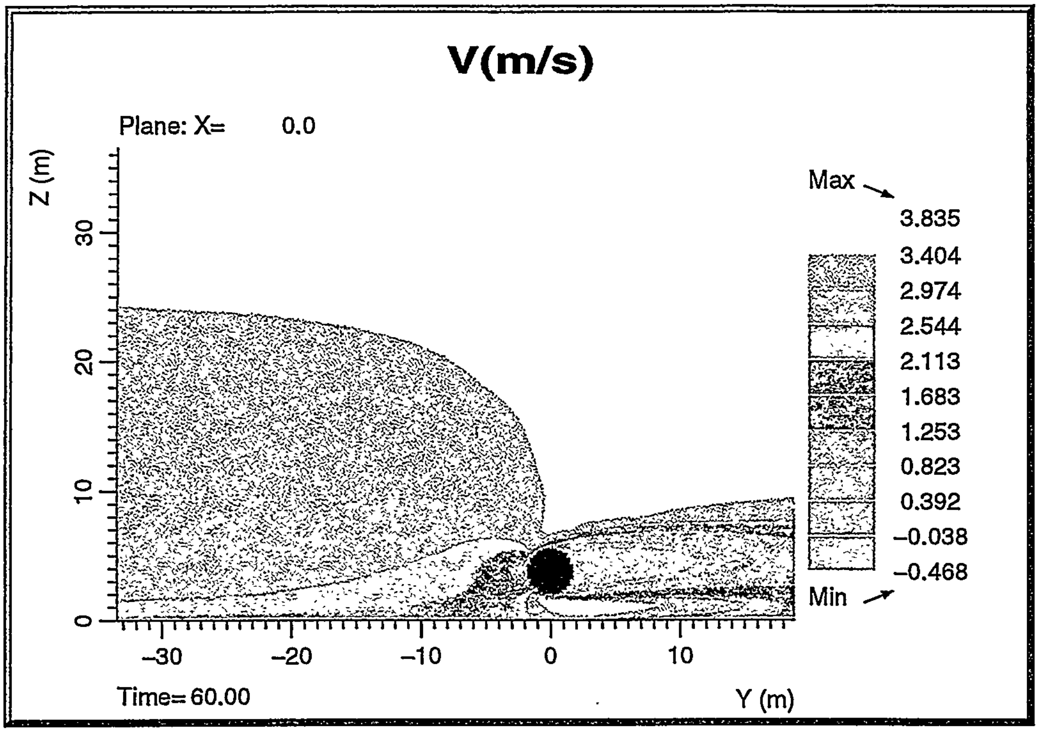


Figure 13. Velocity Distribution Around Cylinder in Cross-Flow, With Ground Plane and Atmospheric Wind Profile: Re=8×10^5

note the effect of the fuselage on the upstream flow and the area above the fuselage. The influence of the fuselage is felt a substantial distance upstream (roughly 2 diameters). In the vertical direction, the fuselage influences the flow field for 6-7 diameters. These results indicate that considerable effort should be taken to locate instrumentation away from large objects (including the upstream direction, and the vertical direction above the object) if measurements of the undisturbed freestream velocity field are desired.

A calculation was performed to investigate the case where the bottom of the fuselage is physically in contact with the ground plane. The same conditions were used as for the previous calculation (Re=8×10^5), with the same wind profile specified. The calculated pressure profile for this case is shown in Figure 14. The stagnation region extends over the entire bottom upwind quadrant of the cylinder (0 to -90 degrees). This is quite different than the preceding cases, where the location of minimum pressure was seen to exist near +/- 75 degrees. The location of minimum pressure (now only occurring on the top of the cylinder) appears at the same circumferential location.

5.5 Calculations of Pressure Field Around a Fuselage in Wind, Including Fire

Two 3D calculations were performed to investigate a more realistic scenario. For these two calculations, a 40x41x36 grid was used to simulate a domain of 150 m x 150 m x 60 m high. A much coarser representation of the fuselage was necessary for these calculations due to the large extent of the grid needed for the fire calculation. The mock fuselage described in

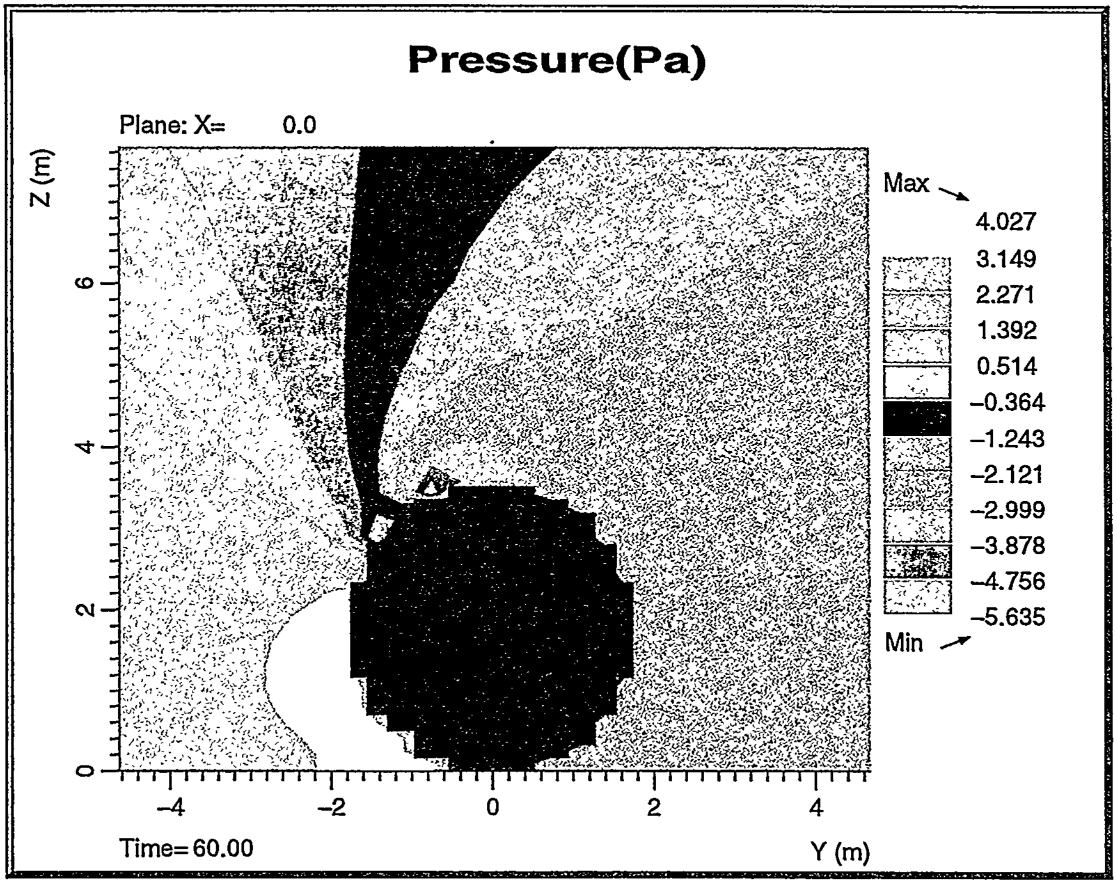


Figure 14. Static Pressure Distribution Around Cylinder On the Ground, and Atmospheric Wind Profile: $Re=8x10^5$

Section 5.2 (a 3.66 m diameter cylindrical section, 20 m in length), was used to simulate an aircraft fuselage located 0.75 m above the ground. The wind speed was specified as 10.2 m/s, at an angle of 22 degrees from the normal to the longitudinal axis of the fuselage (hence the scenario was essentially a cylinder in cross-flow, with a slight longitudinal wind component). The atmospheric wind profile given by Equation 1 was used to specify the distribution of wind speed versus elevation. The fuselage was located near the leeward edge of the pool, which was 18.9 m in diameter, and contained JP8 aviation fuel.

One of the calculations investigated the case where there is only wind, and no fire present. The second case included the effect of fire. Because of the slight longitudinal component of wind, the flame shape varied significantly along the longitudinal axis of the fuselage. The results will be presented at 3 longitudinal locations: X = -9.31 m, -1.07 m, and +7.98 m. The longitudinal component of wind is from the +X direction to the -X direction. Therefore the highest value of +X will have the lowest flame coverage, as the wind sweeps the flames downstream toward the -X position.

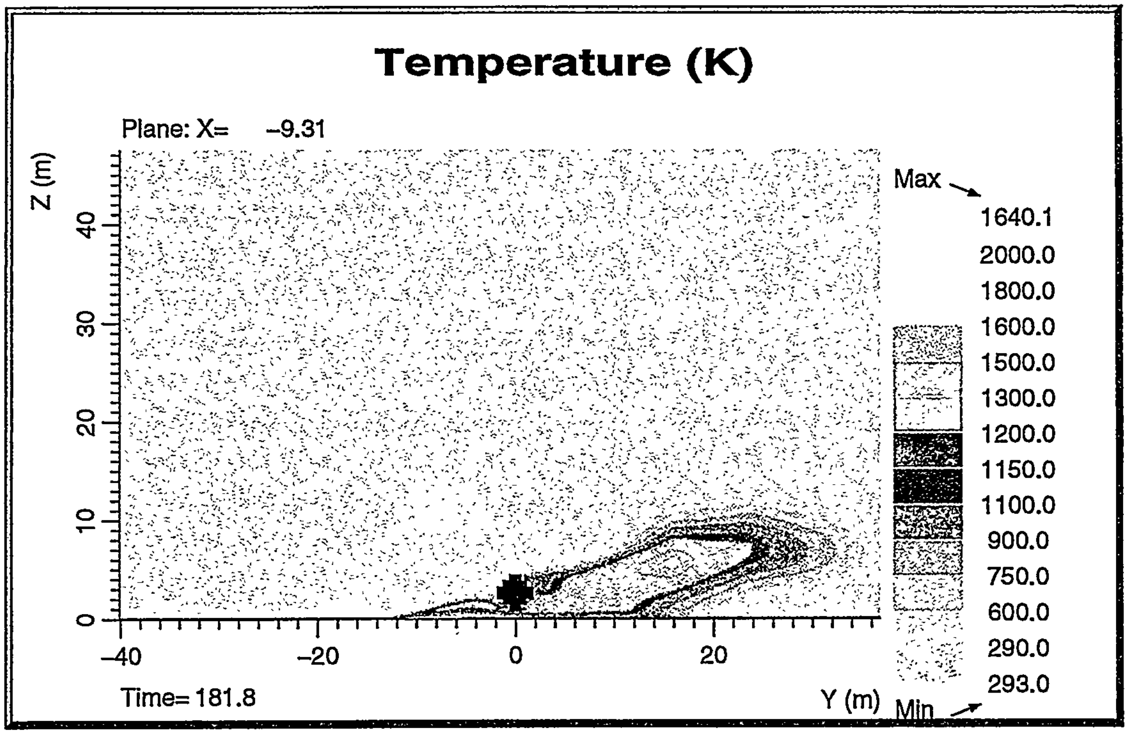


Figure 15. Flame Coverage Around Fuselage in Cross-Wind, at X= -9.31 m

The flame coverage of the fuselage at the X = -9.31 m location is shown in Figure 15, which represents a 2D slice through the diameter of the fuselage. The wind is primarily from the left in the figure (with the small longitudinal component directed out of the figure), and the strength of the wind is such that the majority of the flames are located over the downwind section of the domain, extending for approximately 1 pool diameter downwind. A hot spot is seen at Y = +12 - 18 m, at an elevation of about 5 - 8 m. The wind is strong enough that there is no observable flame coverage of the top upwind quadrant of the fuselage. The velocity field in this cross-section is shown in the top part of Figure 16. For comparison, the velocity field at the same cross-section in the calculations without fire is shown in the bottom part of Figure 16. The lack of flame coverage over the top upwind quadrant leads to relatively similar profiles over that quadrant. The bottom portion of the fuselage also exhibits similar velocity profiles in the two cases, as the flow is tightly squeezed underneath the fuselage and must accelerate independent of the fire. Some difference in velocity field can be noticed upstream of the fuselage, near the ground plane. This difference arises from the thin layer of flame coverage that exists at this location for the case with a fire. The largest differences in velocity field occur over the top and downwind quadrants of the fuselage. The effect of the hot spot downwind of the fuselage is clearly seen, resulting in a local region of significantly higher velocities (~50% larger than for the no fire case). It is

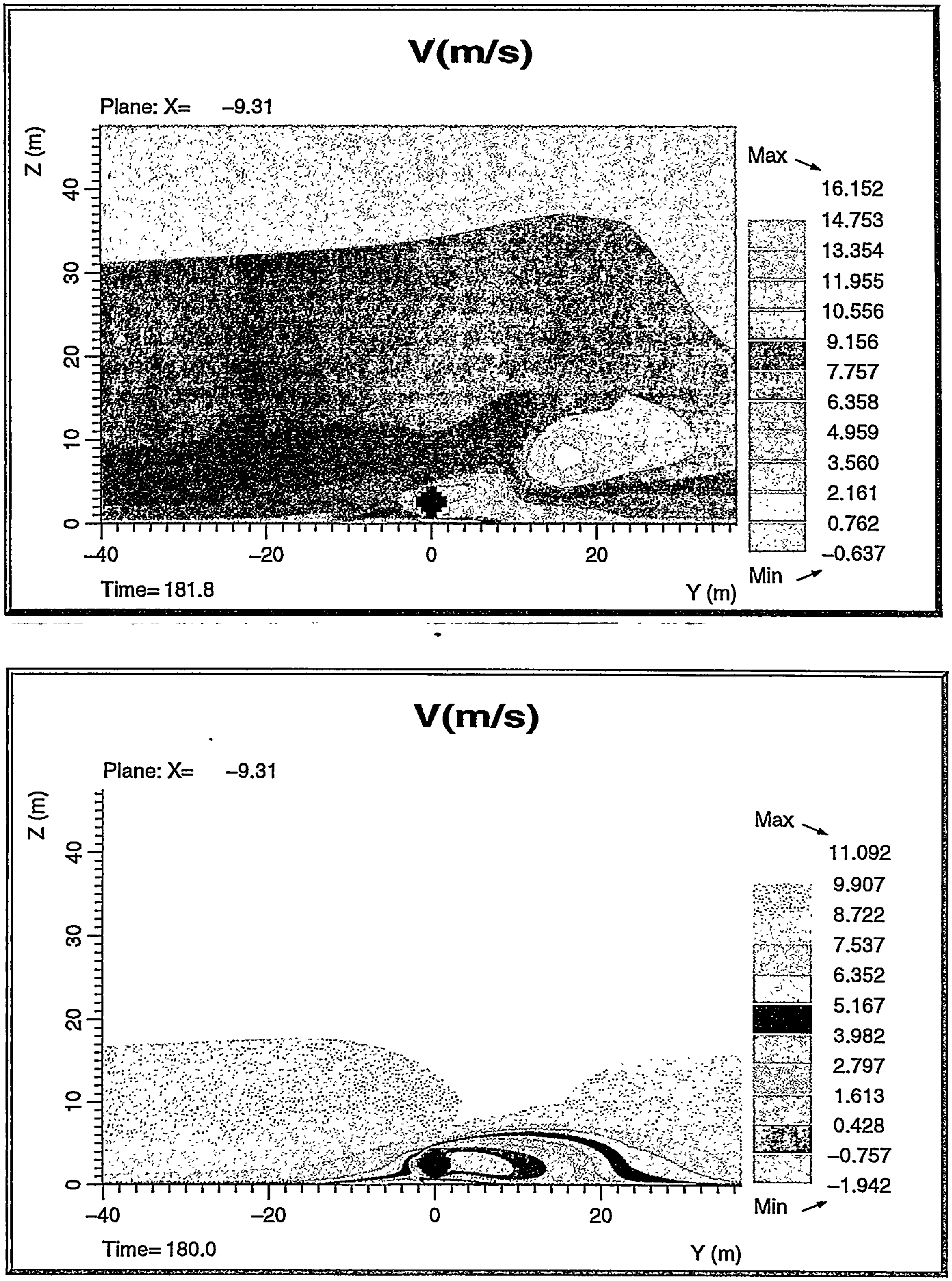


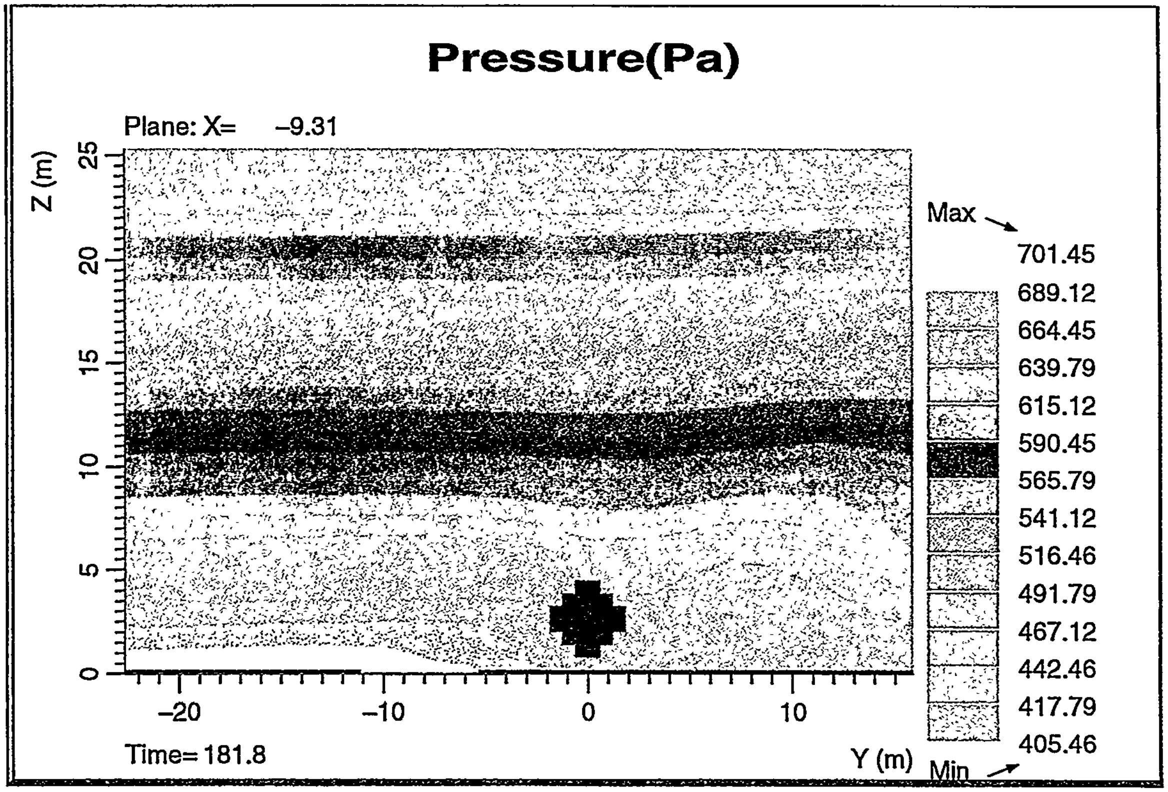
Figure 16. Velocity Field Around Fuselage in Cross-wind.: X= -9.31 m. Top: With Fire Present. Bottom: Without Fire Present.

interesting to note that the recirculation region (wake region) behind the fuselage is much shorter in the fire case, extending only half the distance downwind. Apparently, the vertical momentum field resulting from the buoyancy generated by the fire is strong enough to significantly alter the low momentum wake region downwind of the fuselage.

The pressure fields can also be compared for the two cases. The presentation of pressure for these simulations will be different than for the previous cases without fire. The pressures presented throughout the remainder of this section include the effect of a hydrostatic pressure (necessary for the buoyancy calculations). This makes it more difficult to observe small differences in the pressure field, as the variation in the hydrostatic pressure is greater than that resulting from changes in the velocity field. However, it is the total pressure (i.e., including the hydrostatic pressure) that will determine the penetration of flame and smoke into the fuselage. It should also be noted that only differences in pressure (and not the values of pressure) are meaningful in these results, as the actual values for pressure in the model results are based on an arbitrary reference height for the calculation of the hydrostatic pressure.

The top of Figure 17 shows the calculated pressure field around the fuselage for the case with a fire. For comparison, the bottom of Figure 17 shows the 'no fire' pressure contours. In both figures, near the upstream and top boundaries we see the horizontal bars representative of the hydrostatic pressure distribution and a constant velocity field. In the vicinity of the fuselage, these contours become warped by changes to the velocity field (and therefore, changes in the static pressure). In the bottom of Figure 17, these changes result solely from the presence of the fuselage. In the top of Figure 17, the effect of fire on the pressure field is also included. The differences between the two pressure profiles follow the differences in the velocity fields. There is some difference upstream of the fuselage, near the ground plane (again as a result of the flames being present there). The largest differences are noted to be on the downwind side of the fuselage. For the fire case, a large region with relatively small variation in pressure across it is seen immediately downwind of the fuselage.

For the longitudinal location (along the fuselage) of X = -1.07 m, Figure 18 shows that there is a significant reduction in flame coverage at this location. Flames now extend downwind of the fuselage by about 10 m (versus more than 20 m at the previous location). The upwind flame coverage near the ground is about the same as at the previous location (X= -9.31 m). The velocity field for the fire case is shown in the top of Figure 19, and in the bottom of Figure 19 for the 'no fire' case. Significant differences are again observed on the upwind side of the fuselage near the ground plane. This is expected based on the results at the previous location, as the flame coverage along the ground is about the same at these two longitudinal locations. Velocity profiles underneath the fuselage are similar for the fire and the no fire case. There is still a noticeable difference in the velocity profiles downwind of the fuselage, and over the top portion of the fuselage. As expected, these differences are not as large as those observed at the previous X location, since the flame coverage (and hence, the effect of the fire) is not as great at this location. The pressure profile for the case with fire is shown in the top of Figure 20, and for the 'no fire' case in the bottom of Figure 20. There is not as much difference in profiles upstream along the ground plane as observed at



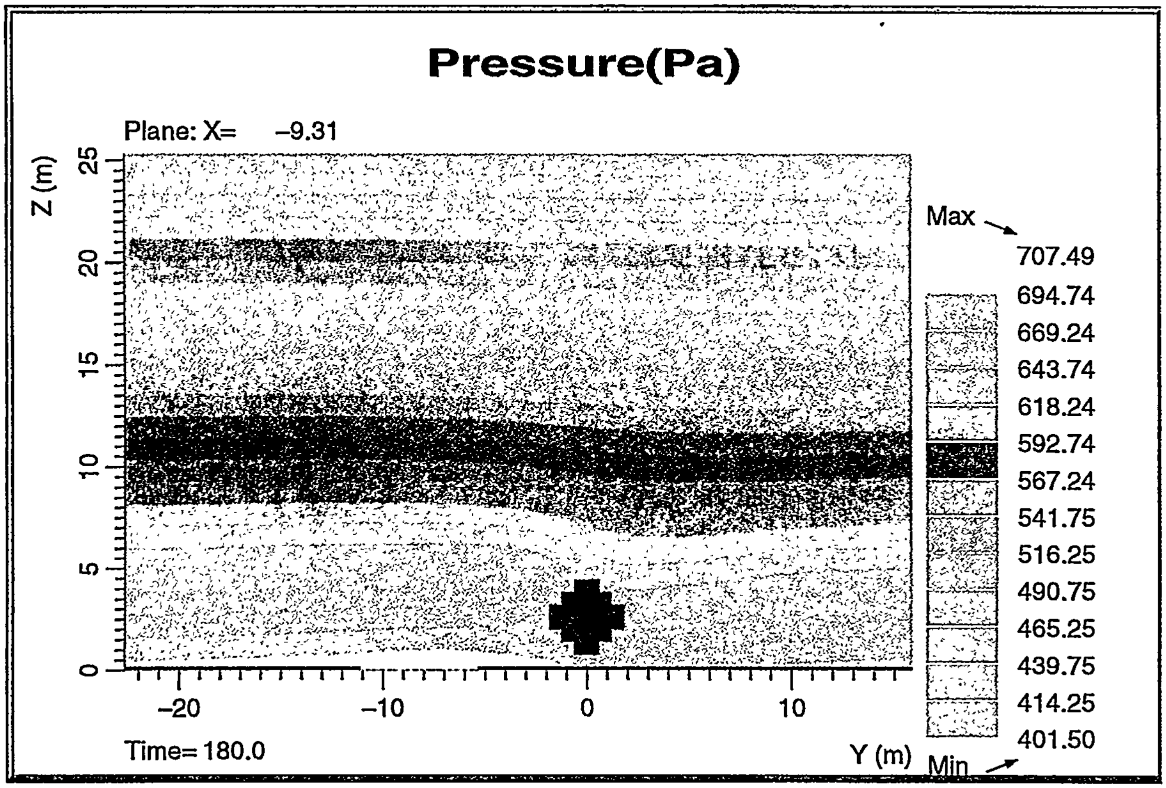


Figure 17. Pressure Field Around Fuselage in Cross-wind.: X= -9.31 m. Top: With Fire Present. Bottom: Without Fire Present.

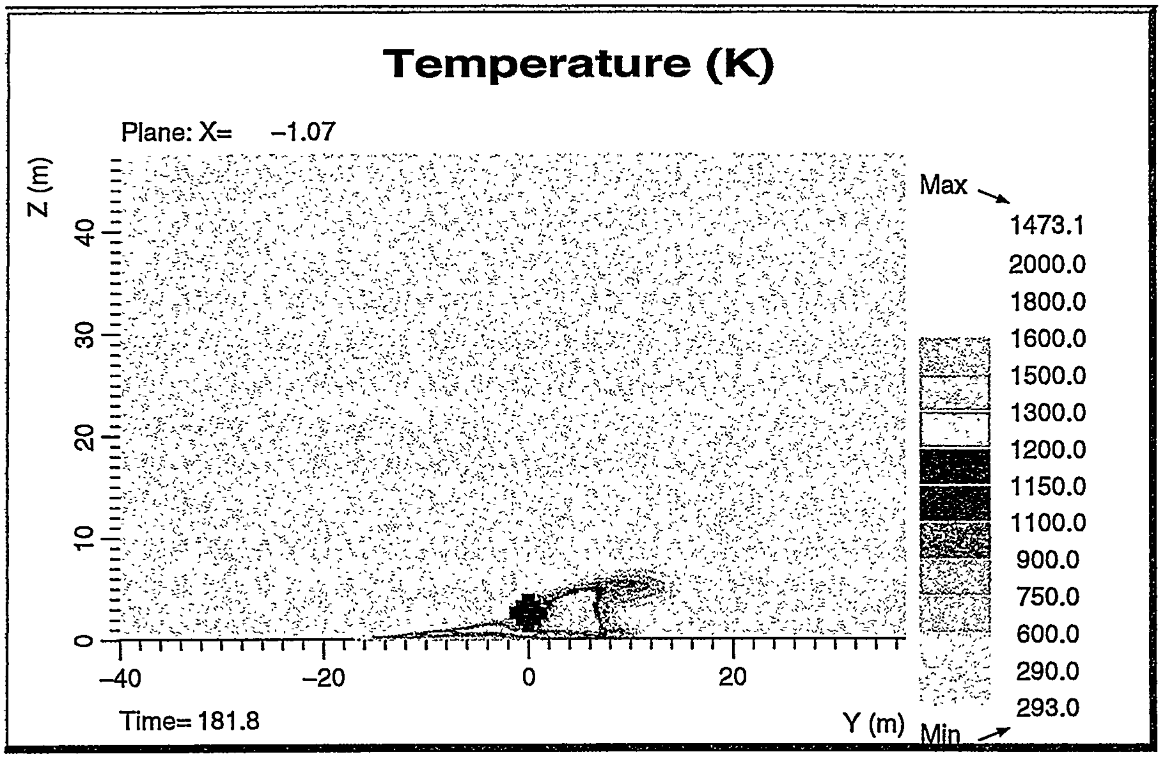


Figure 18. Flame Coverage Around Fuselage in Cross-Flow, at X= -1.07 m

the previous X location, and it is unclear as to why this is the case. Very large differences exist in the downwind pressure profiles, and again for the case with fire a large region is observed immediately downwind of the fuselage which has relatively little variation in pressure across it. The pressure profiles over the top of the fuselage are not very different for the two cases.

At the X = +7.98 m location, there is very little flame present as shown in Figure 21. This is very near the end of the fuselage, and the relatively weak longitudinal component of wind is strong enough to force the vaporized fuel to move toward the pool center. The lack of much of a flame at this location would indicate that the velocity and pressure profiles should be very similar for the case with fire and the 'no fire' case. This is indeed the case. The pressure profiles are shown in Figure 22 for the two cases, and there is very little difference between the two. The velocity fields show a similar resemblance to each other for the two cases, and will not be presented here for brevity.

Due to limited resources, no further calculations were made for this problem. The above results, while preliminary in nature, illustrate the coupling of the fuselage, wind, fire, and ground plane. The pressure and velocity fields are seen to be affected by: the presence of the ground, the location of the fuselage relative to the ground, and the distribution of the wind velocity, as well as by a fire. For the case with a fire, local hot spots in the flame zone result in large velocities that significantly alter the local pressures and distort the

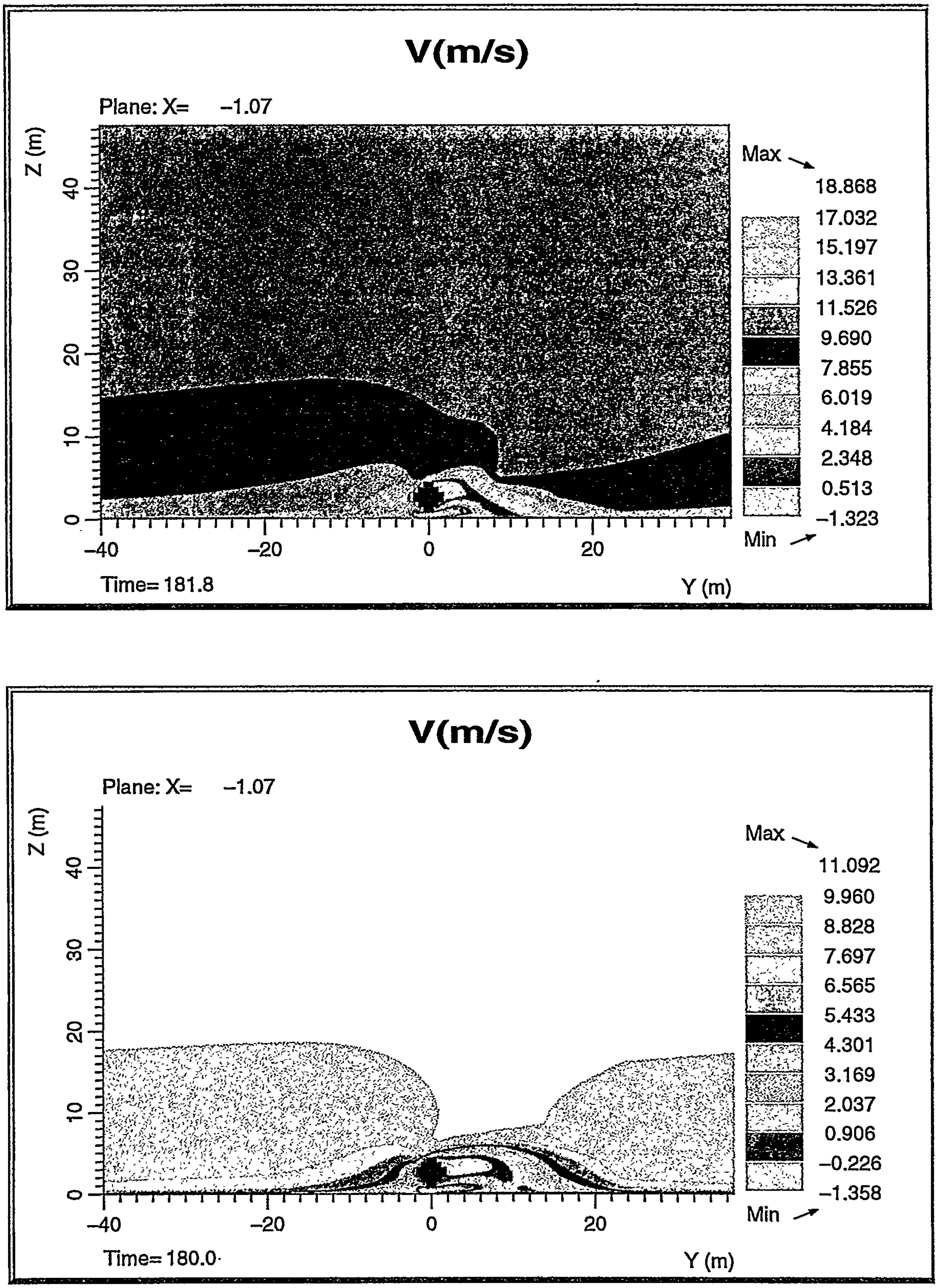
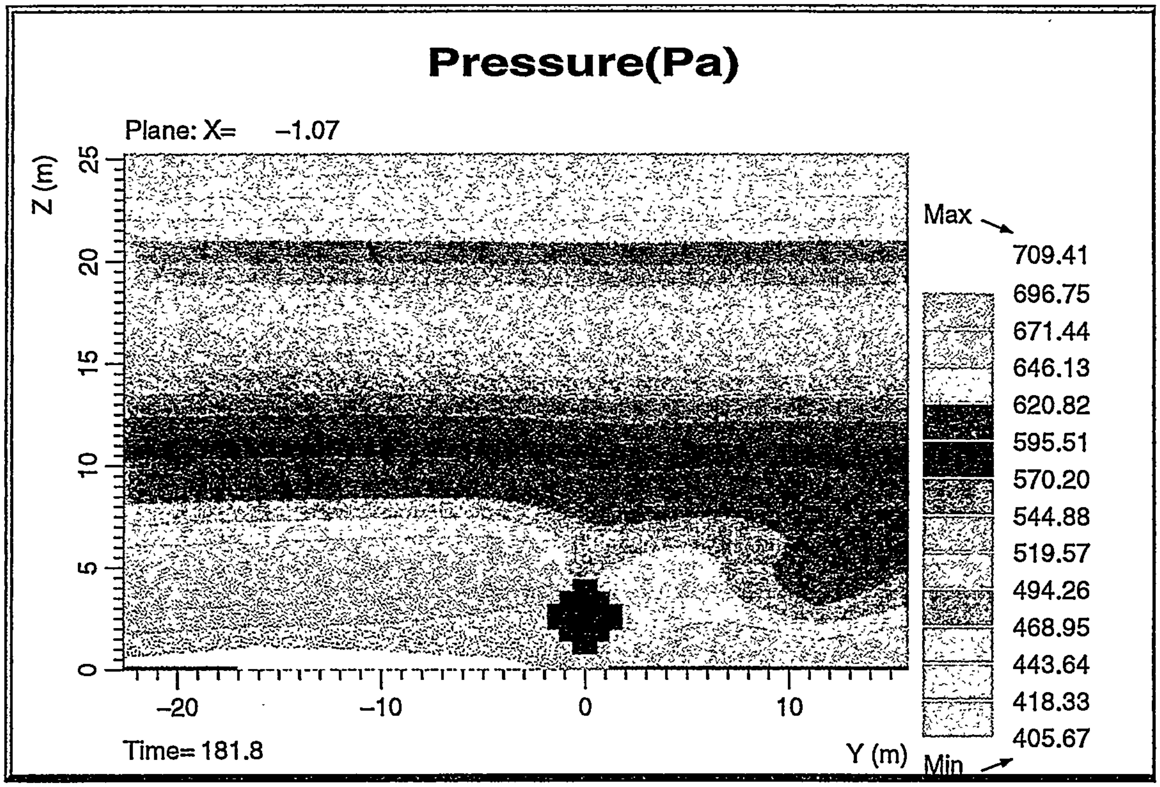


Figure 19. Velocity Field Around Fuselage in Cross-wind.: X= -1.07 m. Top: With Fire Present. Bottom: Without Fire Present.



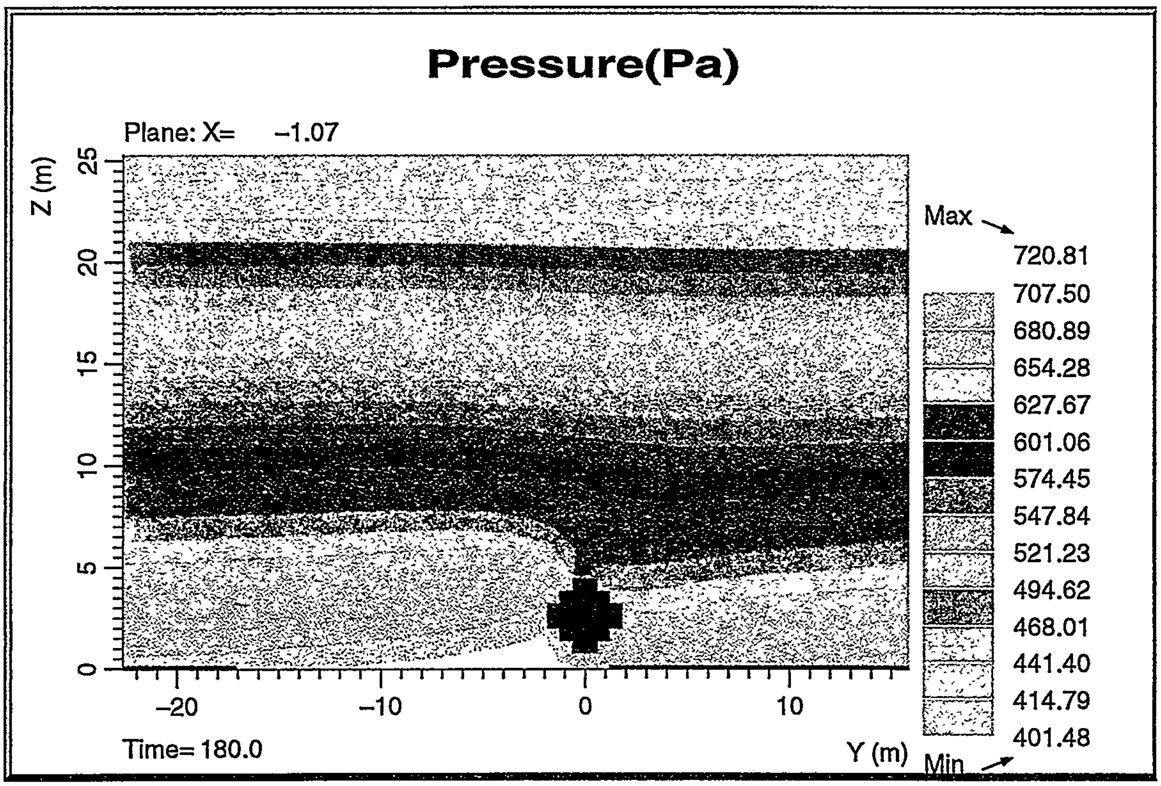


Figure 20. Pressure Field Around Fuselage in Cross-wind.: X= -1.07 m. Top: With Fire Present. Bottom: Without Fire Present.

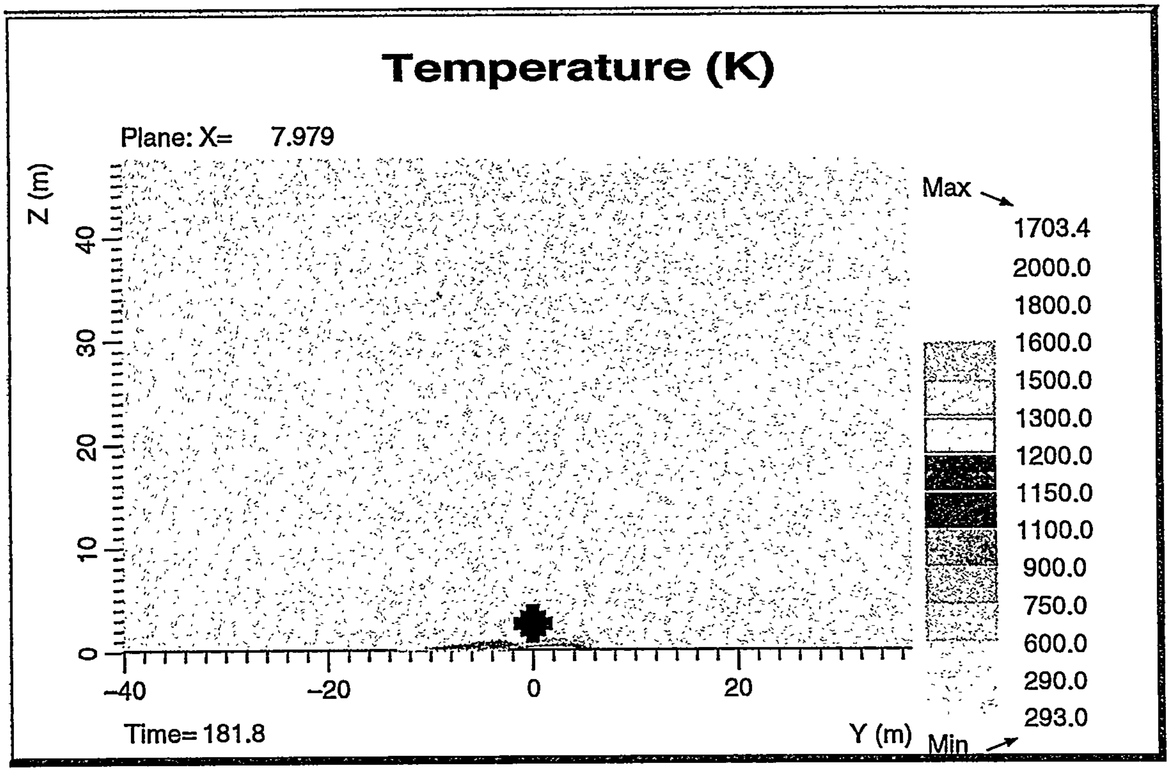


Figure 21. Flame Coverage Around Fuselage in Cross-Flow, at X= +7.98 m

recirculation region downstream of the fuselage. This strong coupling makes *a priori* prediction of flame shape and pressure profile difficult, apart from the use of a computational tool such as VULCAN.

Further research (beginning with good experimental data) is needed to build confidence in the results of such computational tools for predicting these scenarios. The computational results also indicate that considerable effort should be made to locate experimental instrumentation away from large objects such as a fuselage, if measurements of the undisturbed pressure and/or velocity field are desired. This includes the upstream region and the region immediately above the fuselage. Regarding further computational investigation, these results point to the necessity of including substantial amounts of the domain near a fuselage in a cross-wind and fire. This is true not only for external pool fire environments, but also when investigating the impact of wind on an internal fire in an aircraft cabin.

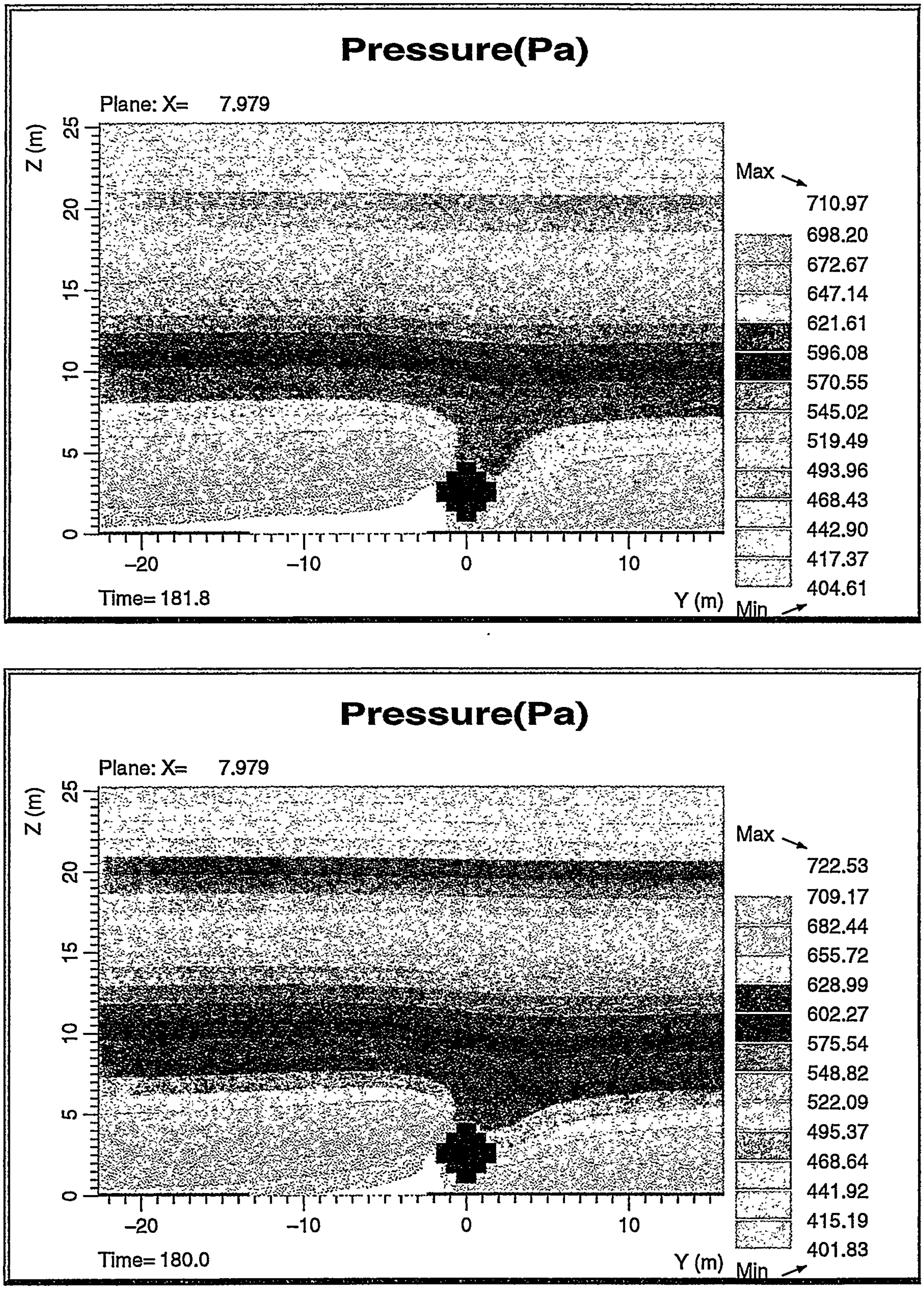


Figure 22. Pressure Field Around Fuselage in Cross-wind.: X= +7.98 m. Top: With Fire Present. Bottom: Without Fire Present.

6.0 Evaluation of Gray Gas Model

Apart from the work devoted to fire field modeling for aircraft fire research, another aspect of this study was to evaluate the accuracy and applicability of simpler fire models for open pool fire environments. In particular, a model previously developed at Sandia (and referred to herein as the Gray Gas Model) was of interest for determining the influence of large objects on the local fire environment. The Gray Gas Model represents a unique type of model, herein referred to as a Risk Assessment Compatible Fire Model, or RACFM. RACFM contain spatially resolved submodels of the dominant physical processes, with empirically-based submodels for the physics of secondary importance. They derive the name RACFM because they are generally fast-running models, making them well-suited for use in probabilistic risk assessments. Such models are of interest to FAA because they can be used to better assess the risk that fires pose to aircraft safety.

6.1 Background on Gray Gas Model

A simple analytical model of the interaction between a large object and an engulfing fire environment has been developed previously (Nicolette and Larson, (1990)). In that work, a one-dimensional thermal radiation model (a two-flux model) was coupled to the local flow, thermal, and combustion fields to characterize the interaction between an object and a fire. Many simplifying assumptions were used to derive the model. For example, the combustion products are treated as a gray gas with uniform velocity and freestream temperature away from the object (Figure 23). The model results demonstrated that a substantial thermal radiation boundary layer can develop adjacent to a large, cold object in a fire. Calculations with the model indicated that reductions in heat flux of 25-40% (relative to uncoupled blackbody heat flux calculations) were typical for a large flat plate in a fire, as a result of the thermal radiation boundary layer development. These results have successfully explained discrepancies observed in heat flux measurements to calorimeters of varying sizes (see for example: Keltner et al., (1990)). This object/fire interaction could be important for aircraft fires involving fuel tank heating, or for very short times after ignition (e.g., the first few minutes in which burn-through and evacuation occur). Each scenario of concern should be evaluated on a case by case basis.

A second paper regarding this model (Gritzo and Nicolette, (1996)) built upon the previous work, but went further in adding the transient thermal response of a large object to the parameter space. In that work, the governing equations were non-dimensionalized and solved over a broad parameter space. The non-dimensional heat flux was found to be a function of a radiation number (N_{rad}), and a radiation Biot number, and the ratio of object temperature to fire temperature. The results were then utilized to delineate regimes in which the fire/object interaction was important, and others where it could be neglected. Also, a matrix of results in non-dimensional form was generated for design use.

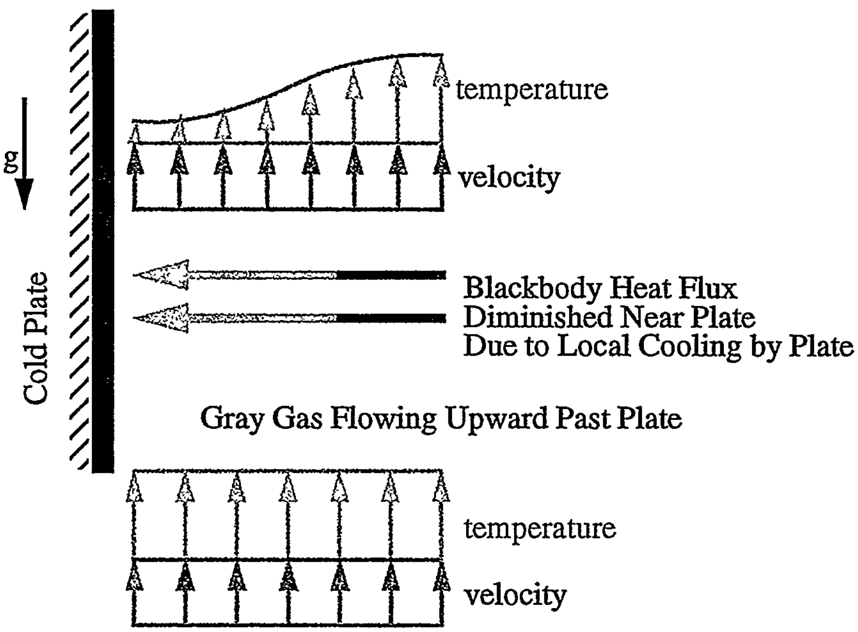


Figure 23. Gray Gas Model Schematic Representing Radiation/Convection Interaction Between a Heated Gray Gas and a Cold Flat Plate

6.2 Comparison of Gray Gas Model to VULCAN

The purpose of the present study was to investigate the validity of the results of the earlier studies in light of the many assumptions that were invoked in the development of the Gray Gas Model. The results of VULCAN calculations for a vertical flat plate in a fire were used for the comparison. Since the Gray Gas Model has been described in other references (Nicolette and Larson (1990), Gritzo and Nicolette (1995)), it will not be described further herein. The interested reader is referred to those previous publications if more details are desired.

To assess the validity of the simplifying assumptions which are inherent in the Gray Gas Model, two cases were considered. In order to simplify the comparison, the transient response of the plate was not considered in either case. For Case 1, the heat flux to a vertical, constant temperature flat plate with a heated gray gas flowing past it was considered. This problem was selected because it allows for a direct comparison to be made, without complicating the comparison due to combustion and soot effects.

First, both VULCAN and the Gray Gas Model were benchmarked against a 1D thermal radiation diffusion solution, to ensure that the radiative transport algorithms were accurate. Both models agreed with the textbook solution to within 0.1%.

For Case 1, 2D calculations were performed for a 3-5 m long (*L*) vertical plate at 300 K. The freestream temperature of the gray gas flowing past the plate was 1300 - 1373 K. The freestream velocity of the gases was 5 m/s, and the absorption coefficient ('a') was specified as 1 - 5 m^{-1}. This corresponds to a range of radiation number given by:

$$0.01 < aL \times N_{rad} < 2.0 \tag{2}$$

where $N_{rad} = \frac{\sigma T_f^3}{\rho_f C_p V_f}$, σ is the Stefan-Boltzmann constant, T_f is the freestream flame temperature, ρ_f and C_p are the fluid density and specific heat, and V_f is freestream velocity.

An initial calculation with the field model was performed using the same boundary conditions and assumptions inherent in the Gray Gas Model, including a uniform absorption coefficient. The leading edge and far-field fluid temperatures were specified boundary conditions for the problem, along with a specified uniform velocity profile. The results were in excellent agreement (<1%). Then the assumptions in the field model were relaxed one by one, and the results of the two models were compared. The assumptions in the field model were relaxed such that the field model calculation included the effects of: variable properties, buoyancy, wall drag, turbulence, and 2-dimensional velocity and radiation fields. For all of the assumptions investigated, the Gray Gas Model-predicted temperature field was in very good agreement with that predicted by the field model. The quantity of direct interest in most fire calculations is the heat flux to an object. For the range of conditions analyzed, the agreement between the Gray Gas Model heat flux and the fire field model heat flux was within 5% (for radiative heat fluxes averaged along the length of the plate), as shown in Table 1. This demonstrates that the assumptions inherent in the Gray

Table 1: Case 1 Heat Flux Agreement

$aL*N_{rad}$	Q_{RACFM} (kW/m^2)	Q_{VULCAN} (kW/m^2)	% difference
0.01	196	191	+2.6
0.1	176	177	-0.01
0.3	161	153	+5.2
2.0	80	78	+2.6

Gas Model are very good for this particular problem. Therefore, in spite of its simplicity (relative to the field model), the Gray Gas Model is very capable of accurately calculating the heat flux to a plate under these conditions. It was also noted that similar agreement was obtained for the *local* heat fluxes at various locations along the flat plate.

But a more relevant question is: How well will the Gray Gas Model calculate the heat flux to a plate in a fire with local combustion and soot effects (as opposed to the case of a heated

Temperature (K)

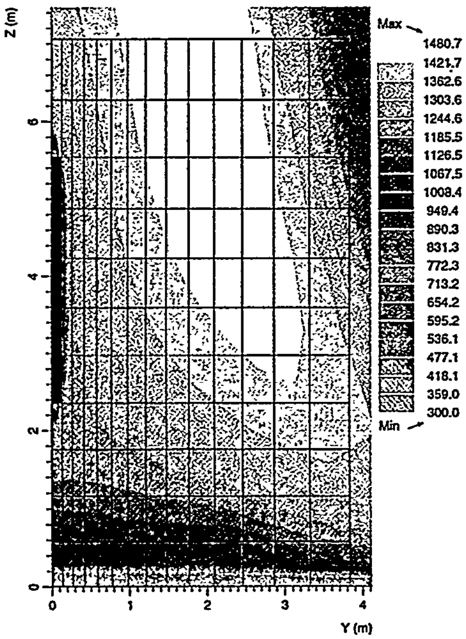


Figure 24. VULCAN-Calculated Temperature Field Near Flat Plate in Pool Fire

gray gas flowing by the plate)? To address this question, Case 2 was considered. Case 2 compared the Gray Gas Model heat fluxes to the results of the fire field model for a 3D calculation of a large pool fire with a large vertical flat plate near the center of the pool, in zero wind conditions. A 16 m diameter pool of JP4 was considered, with a 3 m high plate that was elevated 2 m above the pool and maintained at 300 K. The temperature field calculated by the field model is shown in Figure 24, and the velocity field near the plate is shown in Figure 25.

As seen in Figures 24-25, a direct comparison of the field model results to the Gray Gas Model results is difficult because of the non-uniformity in the freestream gas temperature and velocity in the field model calculations. These variations result from the strong coupling of the physical phenomena which comprise a fire. Large local variations in the quantities of interest (e.g., temperature, velocity) occur as a result of the strong coupling of the combustion, radiation, turbulence, buoyancy, and momentum. This coupling is built into the equation set of VULCAN, and it therefore predicts that large local variations exist in the quantities of interest. Such strong coupling of all the physical quantities is not represented in the Gray Gas Model, and hence such large local variations do not appear in the Gray Gas Model results. Also, some of the non-uniformity is due to the 3D nature of the field model results, versus the 2D RACFM. Therefore, direct comparison of results between the two models is difficult. However, if one accepts the differences between the two models, some comparison is possible, as shown below.

Vertical Velocity (m/s)

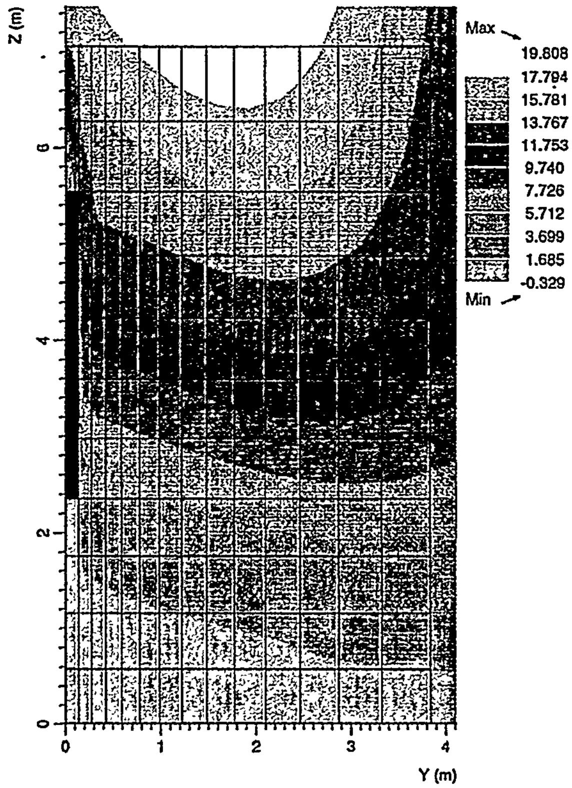


Figure 25. VULCAN-Calculated Vertical Velocity Field Near Flat Plate in Pool Fire

In order to make a comparison between the two models, it must first be noted that the Gray Gas Model is presently set up to output the ratio of the reduced heat flux to a cold object relative to the blackbody heat flux. Therefore one must compare ratios of heat flux, rather than heat flux itself. The Gray Gas Model was developed to assess the reduction in heat flux (from the black body heat flux) that a large, cold object induces in the local fire environment. For the Gray Gas Model, the blackbody heat flux is defined by the user at the far-field boundary by specifying a freestream temperature. For VULCAN, the definition of an appropriate black body heat flux is not as clear, as temperatures and absorption/emission coefficients vary throughout the domain. In order to obtain an appropriate baseline (i.e., blackbody heat flux) for the field model results, another field model calculation was performed without the cold plate. The resulting heat flux to the vertical centerline of the pool was then taken as the baseline heat flux for assessing the reduction in heat flux which the large cold plate induced. This is referred to below as 'Q_{3D} without the plate present.' A comparison of the VULCAN-calculated temperature field for the cases with and without a flat plate in the fire is shown in Figure 26.

A comparison of the results (Table 2) indicates that the Gray Gas Model heat fluxes (averaged along the plate) are higher near the leading edge by 16%, and lower near the

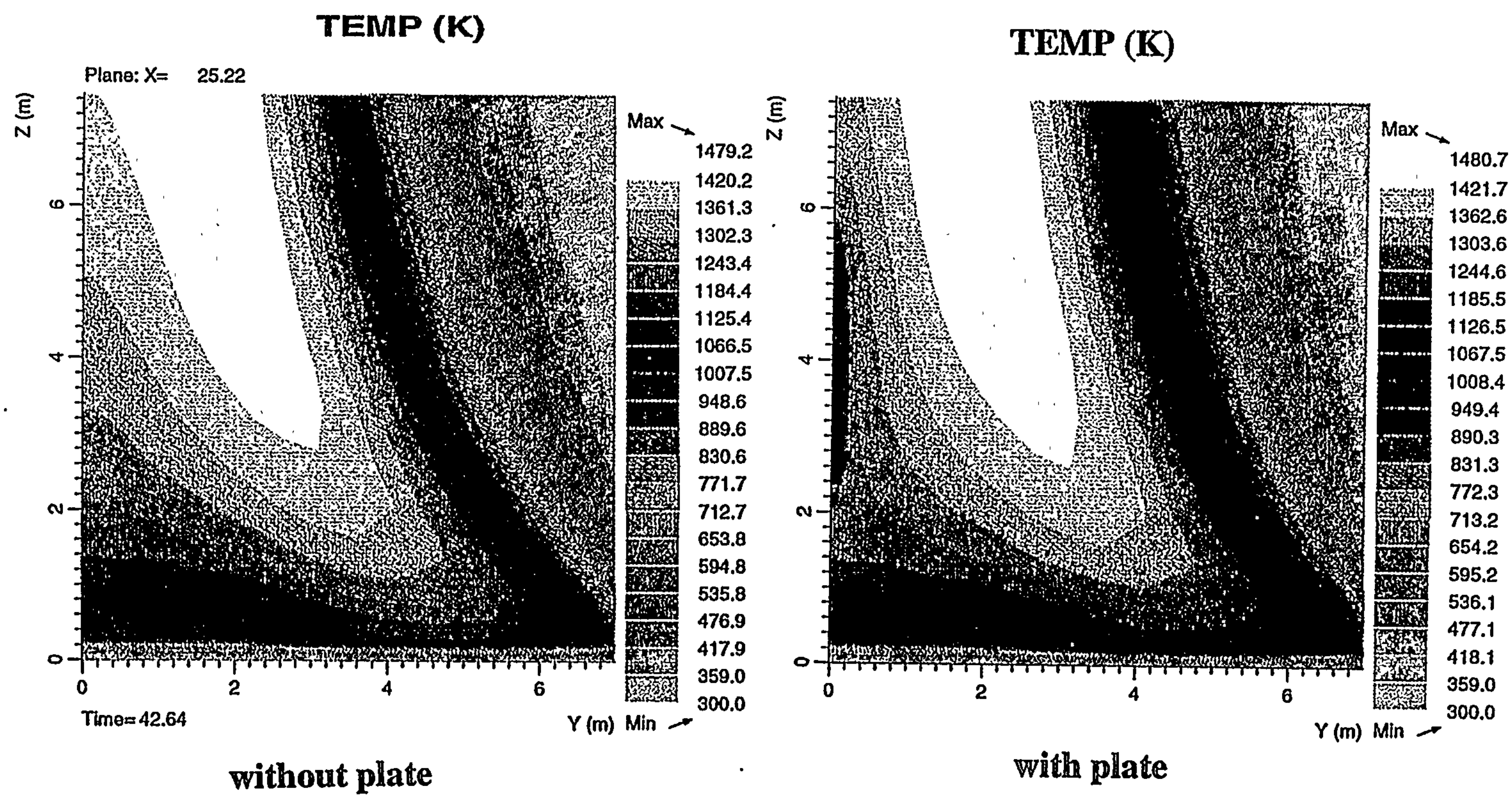


Figure 26. VULCAN-Calculated Temperature Field in Pool Fire. Left: Without Plate. Right: With Plate

Table 2: Case 2 Heat Flux Comparison[a]

Distance along plate	aL^*N_{rad}	T_{rad-3D} [b]	Q_{3D} [c] (kW/m^2)	Q_{RACFM} [d] (kW/m^2)	% difference
0	0	1290	124	144	+16
.5	.14	1307	124	129	+4
1.	.28	1322	124	126	+3
1.5	.38	1335	125	125	0
2.	.48	1346	126	123	-3
2.5	.58	1356	127	121	-4
3.	.65	1361	128	120	-6

a. Columns 2 - 5 are averaged along the plate over a distance L.
b. T_{rad-3D} is the gas temperature from the 3D field model results without the plate.
c. The subscript '3D' indicates the 3D field model results.
d. Q_{RACFM} is defined as Q_{3D} without the plate present multiplied by: $(q/\sigma T_f^4)_{RACFM}$ where q is the Gray Gas model heat flux, and σT_f^4 is the blackbody heat flux based on a freestream flame temperature (T_f).

trailing edge by 6%, relative to the fire field model results. This agreement is considered to be quite good, given the assumptions inherent in the Gray Gas Model. The largest discrepancy occurs near the leading edge, due to the 2-dimensionality of the radiation field there. Also, some discrepancy occurs because the Gray Gas Model utilizes a uniform freestream temperature, whereas the field model results indicate significant variation in the freestream temperature field due to the local combustion process.

The field model heat fluxes shown in Table 2 suggest that the heat flux to the plate is relatively uniform. However, the third column shows that there is a significant increase in the freestream temperature of 71 K, which should lead to a 24% increase in heat flux to the plate near the trailing edge. This effect is counter-balanced by the gray gas effect, which tends to reduce the heat flux to the plate because of local cooling due to the plate being maintained at 300 K. The net effect is that the heat flux to the plate for this particular scenario is, coincidentally, relatively uniform.

This comparison has been very helpful in assessing the level of accuracy of the Gray Gas Model. The results indicate that the Gray Gas Model does indeed capture the dominant physical processes of the problem of interest. It can therefore be used with confidence to estimate the influence of large, cold objects on the local fire environment. The above results also demonstrate the utility of a fire field model such as VULCAN for RACFM validation and development. By turning different submodels on and off, the importance of different assumptions in the RACFM can be evaluated. This feature could also be of value in developing RACFM for aircraft fire scenarios, as it indicates the impact (or lack thereof) of each of the physical submodels upon the result of interest for a particular problem.

Intentionally Left Blank

7.0 Summary

As part of the Aviation Safety Act of 1988, the FAA was mandated to perform long-range research that will lead to new technologies for improving aircraft fire safety. In response to this mandate, the FAA created a Fire Research Branch and developed a Fire Research Program. Fire modeling was identified as one of six important research areas which must be addressed to reach the goals of the program. Sandia assisted FAA with development of the Fire Modeling thrust area via an Inter-agency Agreement (IA DTFA03-93-X-00014). This report has documented work done by Sandia for the FAA in support of this agreement.

To initiate the Fire Modeling thrust work, a structured, long-range fire modeling plan was developed for aircraft fire research. The first step in this development was the identification of the technical issues relevant to aircraft fire modeling research, along with technical methods for addressing these issues. Fire field models were noted to have significant potential for addressing the technical issues of interest because of their spatial and temporal resolution of the fire environment, both for external pool fires and internal cabin fires. RACFM were also identified to have potential for application to aircraft fire problems, especially for risk assessment purposes. For fire field models, the major technical questions in need of further research are: turbulence modeling, soot generation and agglomeration, and radiative properties of soot-laden gases. An additional concern (applicable to all types of fire models, not only field) is the sensitivity of the fire model. This includes: sensitivity to input parameters, and sensitivity to modeling assumptions (including numerical scheme). An overall sensitivity analysis was identified to be of great benefit to a results-oriented, long-range fire modeling research program, and may be the best framework from which to structure an overall approach to fire modeling research. Also, the need for validation of fire models, in terms of the data needed, and the interpretation of the data and the model results, was observed to be critical for further development of fire models.

Based on these results, a long-range research plan was drafted for the FAA Fire Modeling thrust area. The purpose of this plan was to define a tactical means for achieving the strategic goals of the Fire Modeling thrust area, which were: the development of analytical tools for aircraft fire modeling, and the integration of the other thrust areas into a coherent long-range aircraft fire research program. Although the research plan was never finalized due to a reduction/re-direction of funding, research was initiated in several of the fire modeling research areas. This research has been summarized herein.

Existing fire field models were reviewed, and the most promising model was selected for further evaluation. The VULCAN fire field model was identified as one of the most promising models, and an evaluation of its capabilities relative to fires involving aircraft was initiated.

The evaluation began with a 3D simulation of a pool fire involving an aircraft with a cross-wind. This calculation successfully demonstrated the temporal and spatial resolution that can be obtained using a field model for aircraft fire problems. The tendency of the fire to attach itself to the fuselage was observed in the calculations. Also, the sensitivity of the model results to the extent of the computational domain was noted. In particular, computational domains that were too small resulted in non-physical flow fields. When the computational domain is made sufficiently large, such problems are avoided.

Next, a 3D calculation was made to compare the field model results to the results of a large-scale test involving a mock aircraft fuselage in a pool fire and a cross-wind. Reasonable agreement was obtained for the radiative heat flux to the fuselage. Heat fluxes to the pool were calculated to be somewhat lower than measured. Large vortical structures were observed in the calculated results, in agreement with the structures observed during the test. These structures are believed to be responsible for the large spatial variations in heat fluxes to objects in large pool fires. The field model results were observed to be very sensitive to the height of the belly of the fuselage above the pool, as a result of the high cross-wind. The model results clearly demonstrate a tight coupling of the fire, wind, and fuselage. This is to be expected, due to the tight coupling of the actual physics.

A large-eddy simulation capability for fire was developed. The LES submodel allows for puffing motions of the fire to be simulated. Such transient turbulent structures may be important for prediction of such phenomena as remote ignition, flame spread, and pressure distribution around a fuselage. Good agreement was observed between the calculated and experimentally measured puffing frequencies of pool fires over a range of pool diameters. A demonstration calculation involving a mock fuselage was also performed. The results lend insight into the transient nature of the vortical structures that form due to the interaction of the fire, wind, and fuselage. Large spatial and temporal variations in heat flux to the fuselage were calculated.

A limited number of 2D calculations of the static pressure and velocity field around a fuselage in a cross-wind were performed (without a fire). It is very difficult to make such calculations, and the reasons why are summarized. The calculations were performed without a fire present to establish a baseline for comparison to the case with a fire, and to evaluate the success of the model to predict flow and pressure fields. A calculation was performed for a cylinder in cross-flow (without a ground plane). Very good qualitative and quantitative agreement is obtained (relative to published results) for the shape of the pressure distribution around the fuselage (cylinder), and the location of the stagnation and separation points, and location of minimum pressure. The influence of the ground plane was investigated, along with grid refinement. Qualitative agreement with previously published results is obtained for the static pressure on the top side of the fuselage relative to that on the bottom. The effect of including an atmospheric boundary layer profile for the wind velocity distribution near the ground is investigated. There is significant impact on the magnitude of the pressure at stagnation and near separation, as well as in the extent of the wake region downstream of the fuselage. One calculation was also performed with the fuselage belly in contact with the ground plane. Significant differences are noted on the upwind side of the fuselage.

A 3-dimensional calculation was performed for the case of a fuselage in a cross-wind, with a fire. The presence of local hot spots in the flame zone significantly alter the local velocity and pressure distributions compared to the 'no fire' case. For the case with a fire, there is a large region of relatively constant pressure immediately downstream of the fuselage. The length of the recirculation region downstream of the fuselage is about one half as large as in the 'no fire' case. While the pressure distribution results are preliminary in nature, they provide many insights into the flow and pressure fields that occur around fuselages in a cross-wind, with and without fire. This work cites the need for obtaining good experimental data for validation of the model predictions, before the model can be used with confidence in this arena. The model results also indicate that it is important to locate instrumentation far away from large objects such as fuselages, for measurements of the undisturbed flow and pressure fields.

In addition to fire field models, RACFM were also identified to have significant potential for addressing FAA fire issues. A study was performed with VULCAN to investigate the effect of the simplifying assumptions in the SNL Gray Gas Model upon the model results. Calculations were performed with VULCAN and compared to the Gray Gas Model results for 2 scenarios. For the first scenario, a vertical flat plate immersed in a gray gas, the results were within 5%, demonstrating the appropriateness of the assumptions inherent in the Gray Gas Model. The second scenario considered a vertical flat plate in a calculated pool fire environment with zero wind. The results indicate that the Gray Gas Model heat fluxes (averaged along the plate) are higher near the leading edge by 16%, and lower near the trailing edge by 6%, relative to the VULCAN results. This agreement is considered to be quite good, given the assumptions inherent in the Gray Gas Model. The largest discrepancy occurs near the leading edge, due to the 2-dimensionality of the radiation field there. Also, some discrepancy occurs because the Gray Gas Model utilizes a uniform freestream temperature, whereas the field model results indicate significant variation of the freestream temperature field due to the local combustion process.

In conclusion, calculations performed as part of this work successfully demonstrate the significant potential that fire modeling has for aircraft fire research. Computational fire modeling tools can provide both qualitative and quantitative insights into the complex problem of fires involving aircraft.

Intentionally Left Blank

8.0 References

Burns, A. D., Ingram, D., Jones, I. P., Kightley, J. R., Lo, S., and Wilkes, N. S., FLOW3D: The development and application of release 2, Harwell Laboratory, AERE-R 12693 (1987).

Cetegen, B., and T.A. Ahmed, Experiments on the Periodic Instability of Buoyant Plumes and Pool Fires, Combustion and Flame, 93:157-184 (1993).

Eklund, T.I., "Fire Research Plan," Federal Aviation Administration, U.S. Dept. of Transportation, FAA Technical Center, Atlantic City Intl Airport, NJ, January (1993).

Galea, E.R., and N.A. Hoffmann, The Numerical Simulation of Aircraft Cabin Fires, CAA paper 95008, Fire Safety Engr. Group, Center for Numerical Modelling and Process Analysis, The University of Greenwich, London (1995).

Gerhart, P.M., and R.J. Gross, Fundamentals of Fluid Mechanics, First edition, Addison Wesley Publishing Co., (1985).

Gregory, J. J., Mata, R., and Keltner, N. R., Thermal Measurements in a Series of Large Pool Fires, Sandia National Laboratories, SAND85-0196, August (1987).

Gritzo, L. A., and Nicolette, V. F., Coupled Thermal Response of Objects and Participating Media in Fires and Large Combustion Systems, Numerical Heat Transfer: Part A, 28:531-545, (1995).

Gritzo, L.A., Moya, J.L., Nicolette, V.F., and J. Gilliland, Fire Characterization of a Large JP4 Pool Fire and Heat Fluxes Incident on an Adjacent Flat Surface, Proc. Int. Conf. on Fire Research and Engineering, eds. Lund and Angell, September 10-15, Orlando, SFPE, Boston, MA, (1995a).

Gritzo, L.A., Nicolette, V.F., Tieszen, S.R., and J.L. Moya, Heat Transfer to the Surface of Large Pool Fires, Proc. of the Int. Symp. on Trans. Phen. in Comb., July (1995b).

Holen, J., Brostrom, M., and Magnussen, B. F., Finite Difference Calculation of Pool Fires, 23rd Symposium on Combustion, pp. 1677-1683, (1990).

Keltner, N. R., Nicolette, V.F., Brown, N.N., and B.L. Bainbridge, Test Unit Effects on Heat Transfer in Large Fires, Journal of Hazardous Materials, vol. 25, pp. 33-47, (1990).

Kou, H.S., Yang, K.T., and Lloyd, J.R., Turbulent Buoyant Flow and Pressure Variations around an Aircraft Fuselage in a Cross Wind near the Ground, Proc. of the First Int. Symp. Fire Safety Science, (1985).

Magnussen, B. F., Particulate Carbon Formation During Combustion, Plenum Publishing Corporation, (1981).

Magnussen, B. F., The Eddy Dissipation Concept, Proceedings of the Eleventh Task Leaders Meeting, IEA Working Group on Conservation in Combustion, Lund, Sweden, (1989).

Nicolette, V. F., and Larson, D. W., The Influence of Large, Cold Objects on Engulfing Fire Environments, Heat and Mass Transfer in Fires, J. G. Quintiere and L. Y. Cooper (eds.), ASME HTD Vol. 141, pp. 63-70, (1990).

Nicolette, V.F., Gritzo, L.A., Moya, J.L., and S.R. Tieszen, Comparison of Large JP4- and JP8-Fueled Pool Fires, Proc. Int. Conf. on Fire Research and Engineering, eds. Lund and Angell, September 10-15, Orlando, SFPE, Boston, MA, (1995).

Patankar, S. V., and Spalding, D. B., Int. Jou. Heat and Mass Transfer, vol. 15, p. 1787, (1972).

Reeves, J.B., and C.D. MacArthur, Dayton Aircraft Cabin Fire Model, Vol. 1, Basic Mathematical Model, FAA report #: FAA-RD-76-120, I, (1976).

Schlichting, H., Boundary Layer Theory, McGraw-Hill Pub. Co., (1979).

Shah, N. G., The Computation of Radiation Heat Transfer, Ph.D. thesis, University of London, Faculty of Engineering, (1979).

Tesner, P. A., Snegiriova, T. D., and Knorre, V. G., Combustion and Flame, vol. 17, p. 253, (1971).

Tieszen, S.R., Nicolette, V.F., Gritzo, L.A., Holen, J.K., Murray, D., and J.L. Moya, Vortical Structures in Pool Fires: Observation, Speculation, and Simulation, Sandia National Laboratories report, Albuquerque, NM, October (1996).

Appendix A:

Draft Program Plan for Long-Range Aircraft Fire Modeling Research

Draft Plan for

FAA Long-Range Aircraft Fire Modeling Research

1.0 Introduction

This document suggests a tactical plan for the Fire Modeling Thrust of the FAA Fire Research Program. The plan is long-range in scope, consistent with the Aviation Safety Research Act of 1988. This is a draft and has not been finalized.

1.1 Goal #1 of the Fire Modeling Thrust: Development of Analytical Tools

There are two long-range goals of the Fire Modeling Thrust. First, the most direct goal is to develop analytical tools for use: 1) by airframe manufacturers and designers for comparing different aircraft designs, and also for generating correlations and design guidelines for aircraft designers/manufacturers, and 2) in probabilistic risk assessments of aircraft fire safety. These tools will enable aircraft designers/manufacturers to assess the impact of various materials and design configurations on fire safety, and thereby develop aircraft with even higher standards of fire safety.

1.2 Goal #2 of the Fire Modeling Thrust: Integration of Other Thrusts

The second goal of the Fire Modeling Thrust is broader than the first: To integrate the Materials, Suppression and Vulnerability Thrusts into a coherent research program. The Fire Modeling Thrust will assist in achieving this goal by providing a vehicle to interface the materials and suppression research with the vulnerability analysis (Figure 1). In this way, the Fire Research Program can meet the stated objectives of establishing:

- 1) where countermeasures (e.g., improved materials, designs, ventilation, suppression) can be most effective
- 2) what countermeasures have to do to be most effective
- 3) what performance requirements would make interior materials virtually flame-proof.

By integrating all of the thrusts, the Fire Modeling Thrust will help provide direction to (and receive maximum benefit from) the FAA Long-Range Aircraft Fire Research Program. In terms of specifics, the Fire Modeling Thrust will integrate the Materials and Suppression Thrusts into the Vulnerability Thrust by:

Materials Thrust

- 1) Defining the thermal environments which new (and existing) materials must withstand
- 2) Determining the impact which new materials could have on improved fire safety
- 3) Helping define appropriate material testing environments, and the importance of environment variables such as depleted oxygen, convection and radiation heat transfer
- 4) Guiding optimized design

Suppression Thrust

- 1) Determining what form of suppression could be effective on a given fire (e.g., is the fire too hot for water sprays to be effective?)
- 2) Determining how much suppressant will be needed (which is a function of estimated fire size and duration)

Intentionally Left Blank

•3) Modeling the transport of suppressants in the aircraft
•4) Guiding research and development in advanced suppression concepts
•5) Implementing new suppression systems (e.g., location)
•6) Providing design guidance/optimization.

In addition, the Fire Modeling Thrust will provide key information to the Vulnerability Thrust by:

<u>Vulnerability Thrust Area</u>

•1) Providing a quantitative estimate of fire damage for a given scenario
•2) Assessing the ability of a fire to propagate once it has been ignited
•3) Modeling the effectiveness of smoke containment systems
•4) Modeling the transport of CO and other toxic/acid gases for coupling to a toxicology model
•5) Determining system response to fire environments
•6) Quantifying the effect of mitigation strategies.

2.0 Overview of Fire Modeling Thrust Plan

A research and development plan has been developed to facilitate the ability of this program to meet its stated objectives. The plan outlines 3 phases for the Fire Modeling Thrust. Phase 1 consists of research and development leading to a validated 'research code' for modeling aircraft fires. Phase 2 consists of the development of validated 'simulator tools' and Risk Assessment Compatible Fire Models (or RACFM, for short) for use by the aircraft industry. Phase 3 consists of technical support and continued improvement of the 'simulator tools' and RACFM during their application by aircraft designers/manufacturers. Note that portions of the 3 phases may overlap in time, and therefore, it is not necessary that one phase is complete before the next can be started.

The terms 'model' and 'modeling tool' used in this document refer to mathematical models of the physical processes in a fire environment. The term 'research code' used herein denotes a continually evolving computer model for answering questions that are research-oriented, and is intended for use by the developers of the software. The 'simulator tools' are streamlined versions (or equivalents) of the research code that are intended for application to well-defined classes of problems in order to satisfy specific industry needs. They will be developed for selected fuselage damage states, environmental and fuel dispersion conditions, and will include enhanced user interfaces and pre- and post-processors. The RACFM are simplified computer models that require a minimum of CPU time. Thus RACFM are suitable for risk assessment calculations that require a very large number of scenarios to be investigated. They may be empirically based, or simplified versions of more complex models. Both RACFM and simulators will be 'user-friendly.'

Note that several related aspects of fire modeling research are not presently addressed in this plan, such as crash dynamics (fuselage integrity) and fuel dispersion. If so desired, these areas of research can be added in the future. They could provide a deterministic method for specifying the initial conditions for the fire models.

In the following discussion, the 3 phases of the plan are outlined (see Figure 2). As one should expect, the outlined level of detail provided for activities in the latter portion of the program decreases with time.

3.0 Phase 1: Development and Validation of Research Code

The Fire Modeling Thrust will be initiated with research and development leading to a validated research code (Phase 1). This research code will attempt to mathematically model all of the important physical processes in a fire within computational limitations, and will be validated against both phenomenological experiments and large-scale data.

Some of the motivating logic used to define Phase 1 activities is discussed in Appendix AA. Phase 1 has 8 major tasks associated with it:

- Task 1.1 Initiation of Fire Modeling Research
- Task 1.2 Preliminary Validation of Model for Pool Fires
- Task 1.3 Model Sensitivity Study for Pool Fires
- Task 1.4 Preliminary Validation of Model for Internal Fires
- Task 1.5 Model Sensitivity Study for Internal Fires
- Task 1.6 Uncertainty Analysis and Reduction
- Task 1.7 Development of Additional Model Capabilities
- Task 1.8 Final Validation of Research Code

The production of a validated research code is the goal of Phase 1. Each of the above tasks in Phase 1 has subtasks associated with it. Each task and its associated subtasks will now be discussed in detail.

3.1 Task 1.1: Initiation of Fire Modeling Research

The intent of Task 1.1 activities is to identify what capabilities are required in an aircraft fire model, and whether these capabilities are present in state-of-the-art fire models. One or two existing models will be selected for further research and development. The main deliverable for Task 1.1 will be a report documenting the subtask results, and defining the direction of future research.

The subtasks associated with Task 1.1 are:

- Subtask 1.1.1 Define Classes of Fire Problems of Interest for Aircraft Safety
- Subtask 1.1.2 Define Technical Issues in Fire Modeling of Aircraft Fires
- Subtask 1.1.3 Evaluate Existing Fire Models for Capabilities and Development Feasibility
- Subtask 1.1.4 Down-Select of Models

Subtasks 1.1.1 and 1.1.2 lay the foundation for evaluating the capabilities of existing fire models. The first subtask will define the classes of fire problems of interest to FAA, including types of fires (e.g., pool fires, upholstery fires, etc.) and external fuselage conditions (in-flight fires, post-crash fires, etc.). This will identify what capabilities are desirable in a fire model for the class of problems of interest in aircraft fire simulation. Note that the Vulnerability/Risk Thrust Area will define the specific fire scenarios that are important from a probabilistic risk point of view. This emphasizes the need to integrate the fire and materials modeling work into the vulnerability thrust in order to properly assess the risk significance of specific scenarios. The second subtask will define the technical issues that must be overcome to make accurate modeling of aircraft fires a reality. Weaknesses in the present state-of-the-art in fire modeling will be identified, to highlight areas of existing models for close investigation and for further research.

Subtask 1.1.3 is to evaluate existing models for their applicability and development feasibility for aircraft fire modeling. Of the variety of analysis tools that could be applied to the problem of modeling aircraft fires, we have chosen to pursue the research, development, and application of fire field modeling for reasons outlined previously. Initially, Subtask 1.1.3 will be focused on several existing fire models: the latest fire field model under development at Sandia National Laboratories in collaboration with SINTEF/NTH (based on the KAMELEON Fire model developed at SINTEF/NTH), the CFDS (Harwell) FLOW3D model, the CHAM PHOENICS model, the CFDRC model from Computational Fluid Dynamics Research Corporation, the University of Notre Dame Fire Model, and any other fire field models that appear to have potential. These models will be evaluated in view of the classes of fire problems and capabilities of interest to FAA.

It is currently anticipated that 2 or 3 problems will be selected for model evaluation in Subtask 1.1.3. This problem set will include external pool fires as well as fires internal to the fuselage. Comparisons will be made between the different models and against existing experimental data.

At the end of the evaluation of these field models, one model will be selected as a framework for further research, development and application (Subtask 1.1.4). (Note: It is quite possible that two models will be selected, since the modeling of external pool fire environments can require quite different capabilities compared to the modeling of internal fire environments. The following assumes that only 1 model has been selected.) Further research and development is anticipated since we anticipate that there is presently no existing model that can be accurately applied over the broad range of fire problems of interest to FAA. This model will be designated the 'research code.' The selection will be based on technical capabilities, potential for further development, and proprietary issues.

3.2 Task 1.2: Preliminary Validation of Model for Pool Fires

The goal of Task 1.2 activities is to provide a preliminary validation study to ascertain the abilities of the selected model (hereafter referred to simply as 'the model') to match limited existing pool fire data. It will also serve as a benchmark to gauge future progress in model development, and will provide confidence as to the degree to which the important physical processes are approximated by the model.

There are 2 subtasks associated with Task 1.2:

- Subtask 1.2.1 Development of Numerical Thermocouple
- Subtask 1.2.2 Preliminary Validation Against Large-Scale Pool Fire Data

Some of the results of the model evaluation work (Subtask 1.1.3) may serve as a beginning for the preliminary validation. Before the preliminary validation is performed, a numerical model of a thermocouple or virtual sensor will be developed (Subtask 1.2.1) for implementation into the fire model. The purpose of this subtask is to enable a more accurate comparison of calculated model results to thermocouple measurements in a fire, since thermocouple measurements are influenced by local convective heat transfer and radiative heat transfer from a volume of gas having some temperature distribution. Existing large-scale pool fire data will be used for a preliminary validation of the model (Subtask 1.2.2).

3.3 Task 1.3 Model Sensitivity Study for Pool Fires

The main intent of Task 1.3 is to provide a sensitivity study of the selected pool fire model in order to guide the model research and development. A document summarizing this sensitivity study will be the main deliverable for this task.

There are many subtasks associated with Task 1.3. They are listed below to provide an overview of the task activities. A brief discussion of the subtasks follows.

- Subtask 1.3.1 Sensitivity to Pool Fire Input Parameters and Environmental Conditions
  - Subtask 1.3.1.1 Sensitivity to Pool Fire Input Parameters
  - Subtask 1.3.1.2 Sensitivity to Pool Fire Environment Conditions
- Subtask 1.3.2 Sensitivity to Pool Fire Submodel Outputs and Assumptions
  - Subtask 1.3.2.1 Sensitivity to Thermal Radiation Submodel
    - Subtask 1.3.2.1.a Soot Submodel
    - Subtask 1.3.2.1.b Fuel Pool and Fuel Vapor Submodel
    - Subtask 1.3.2.1.c Thermal Radiation Numerical Algorithm
  - Subtask 1.3.2.2 Sensitivity to Turbulence Submodel
    - Subtask 1.3.2.2.a k-ε Turbulence Modeling
    - Subtask 1.3.2.2.b Advanced Turbulence Modeling
  - Subtask 1.3.2.3 Sensitivity to Numerical Solution Algorithm
- Subtask 1.3.3 Documentation of Pool Fire Sensitivity Study Results

A sensitivity study is necessary for a number of reasons: 1) it identifies areas of the present models that most heavily impact the end result, and thereby assists in prioritizing future research, 2) it identifies which (if any) of the physics submodels can be simplified or removed from the model, and 3) an understanding of the sensitivities will allow for a more thorough application of the model to aircraft fire scenarios of interest (since the important parameters/submodels that are identified can then be explored over a range of inputs).

A sensitivity study can be carried out at a variety of levels. For example, it could investigate the sensitivity of the output of the fire model to the input parameters (for example, the influence of pool size on flame heat flux). Or it could be much more detailed, and investigate the sensitivity of a particular process to inherent assumptions (for example, the influence of soot particle size on soot combustion rate). The level to which the sensitivity study will be carried out in this work will be guided by the level of detail which is necessary to identify where further research and development are required to advance the fire model. For example, if a first level sensitivity study indicates that velocity and mixing are unimportant in the calculation of heat flux to an object in a fire, then no deeper sensitivity of the turbulence models will be investigated. If a first level sensitivity study indicates that velocity and mixing have a large influence on heat flux calculations, then the turbulence submodel must be investigated to determine what drives the sensitivities. It is necessary to determine what drives the sensitivities so that these areas can be targeted for further research to better quantify and reduce their uncertainties. A more detailed discussion of 'sensitivity' and 'uncertainty' as used in this document is presented in Appendix AA.

The sensitivity study will investigate pool fire and internal fire environments separately, since they can be fundamentally different from a modeling standpoint (boundary conditions, geometry, fuels, ventilation). However, some of the conclusions drawn from the pool fire study may be useful for the internal fire study (for example, possibly the sensitivity to the numerical solution algorithm).

To begin the pool fire sensitivity study, Subtask 1.3.1 will quantify the extent to which the uncertainty associated with the input parameters and environmental conditions is propagated through the model to influence the end result. The sensitivity of the model results to the various input parameters (e.g., fuel properties) will be quantified in Subtask 1.3.1.1. The impact of post-crash fuel pool sizes and wind effects on the research code results will be quantified for representative post-crash scenarios (Subtask 1.3.1.2). This will define the range of inputs and conditions that need to be addressed in future model research and development for application to large external pool fires.

Subtask 1.3.2 will quantify the sensitivity associated with each submodel for pool fire environments. The level at which sensitivity is investigated will be dependent upon the particular submodel, as discussed above. First, the extent to which the output of each submodel is propagated through the model to influence the end result will be investigated. Then, if it is demonstrated that the research code results are sensitive to a particular submodel output, the sensitivity of that submodel output to the submodel inputs and inherent assumptions will be investigated. The goal is to determine and prioritize which submodels (and in particular, what areas of the submodels) need to be refined in order to improve the predictive capability of the model.

Quantifying the sensitivity associated with the thermal radiation submodel (Subtask 1.3.2.1) for pool fire environments will be a high priority, since thermal radiation is the dominant heat transfer mechanism for large, sooty, external pool fires. In particular, this will include quantifying the sensitivity associated with: 1) the soot model (Subtask 1.3.2.1.a), perhaps extending to the soot generation, combustion, agglomeration, particle size, and composition, 2) the fuel pool model and evaporation (Subtask 1.3.2.1.b), perhaps extending to radiation absorption in fuel pool and vapor, pool grid sizes, convective effects in the pool, vaporization rates, pool composition and boiling temperatures, and 3) the thermal radiation transport algorithm itself (Subtask 1.3.2.1.c).

Quantifying the sensitivity associated with the turbulence model (Subtask 1.3.2.2) is also a high priority, as turbulent mixing is often the rate limiting mechanism in large pool fires. In Subtask 1.3.2.2.a, the assumptions and terms in the k-ε model will be investigated for their impact on sensitivity. In Subtask 1.3.2.2.b, advanced turbulence models (such as Large-Eddy Simulation and Direct Numerical Simulation) may be investigated, as necessary.

The main deliverable for Task 1.3 will be a report documenting the model sensitivity study for pool fires. Since the report represents the culmination of a large amount of effort, Subtask 1.3.3 has been assigned for the preparation of this report.

3.4 Task 1.4 Preliminary Validation of Model for Internal Fires

The goal of Task 1.4 activities is to provide a preliminary validation study to ascertain the abilities of the selected model to match limited existing internal fire data. It will also serve as a benchmark to gauge future progress in model development, and will provide confidence as to the degree to which the important physical processes are approximated by the model for internal fires. There are 2 subtasks associated with Task 1.4:

- Subtask 1.4.1 Implementation of Existing Material Burning Submodels
- Subtask 1.4.2 Preliminary Validation Against Large-Scale Internal Fire Data

Some of the results of the model evaluation work (Subtask 1.1.3) may serve as a beginning for the preliminary validation. Before the preliminary validation is performed for internal fire environments, an existing material burning submodel must be implemented into the fire model (Subtask 1.4.1). This is not to be confused with actually developing a material burning model, which will be addressed later. The phrase 'material burning model (or submodel)' as used in this document refers to a model of the behavior of a material in a fire environment (apart from thermal response), including devolatilization, charring, etc., and the associated energy, smoke, and gas release rates.

Existing large-scale internal fire data will be used for preliminary validation of the model (Subtask 1.4.2). The intent is to quantify the ability of the model to predict the resulting internal fire environment, as well as its ability to model the transport of smoke and toxic gases.

3.5 Task 1.5 Model Sensitivity Study for Internal Fires

Task 1.5 will quantify the model sensitivities for an internal fire environment, according to the following subtasks:

- Subtask 1.5.1 Sensitivity to Internal Fire Input Parameters and Environmental Conditions
  - Subtask 1.5.1.1 Sensitivity to Internal Fire Input Parameters
  - Subtask 1.5.1.2 Sensitivity to Internal Fire Environmental Conditions
- Subtask 1.5.2 Sensitivity to Internal Fire Submodel Output and Assumptions
  - Subtask 1.5.2.1 Sensitivity to Material Burning Submodel
  - Subtask 1.5.2.2 Sensitivity to Convective Heat Transfer Submodel
  - Subtask 1.5.2.3 Sensitivity to Smoke/Species Transport Submodel
  - Subtask 1.5.2.4 Sensitivity to Other Submodels
- Subtask 1.5.3 Documentation of Internal Fire Sensitivity Study Results

To begin the internal fire sensitivity study, Subtask 1.5.1 will quantify the extent to which the uncertainty associated with the input parameters and environmental conditions is propagated through the model to influence the end result. In Subtask 1.5.1.1, the sensitivity of the model results to the various input parameters (e.g., wall properties) will be quantified. The impact of environmental conditions (e.g., geometry and ventilation conditions) on the research code results will be quantified for representative internal fire scenarios (Subtask 1.5.1.2). This will define the range of inputs and conditions that need to be addressed in future model research and development for application to internal fires.

Subtask 1.5.2 will quantify the sensitivity associated with each submodel for internal fire environments. The level at which sensitivity is investigated will be dependent upon the particular submodel, as discussed above. Specific submodels have been identified for the sensitivity analysis, based on assumptions about the dominant physical processes that must be modeled.

Since the material burning submodel calculates the energy and species release rates, the resulting internal fire environment will be tightly coupled to this submodel (Figure 3). Subtask 1.5.2.1 will investigate the sensitivity of the model results to the uncertainty associated with the material burning submodel. This sensitivity analysis will also provide information to the materials research thrust regarding which material parameters have the most impact on the resulting internal fire environment.

Convective heat transfer to walls, objects, and non-ignited fuel may play an important role in characterizing the internal environment. The sensitivity of the calculated internal environment to the convective heat transfer submodel will be assessed in Subtask 1.5.2.2.

Smoke and toxic gases are a major source of lethality in aircraft internal fires. Subtask 1.5.2.3 will investigate the sensitivity of the calculated internal environment to the smoke and species transport submodel.

The other submodels that appear to have large influence on the model results will be investigated in Subtask 1.5.2.4. It is anticipated that the sensitivities of some of the submodels could be much different for an internal fire relative to a pool fire. For example, one might expect different sensitivities to the soot model for external pool and internal fires due to the difference in soot production of a jet fuel pool fire relative to a piece of burning upholstery. Thus, some of the submodels investigated for the pool fire sensitivity study may need to be re-examined for the internal fire environment.

The main deliverable for Task 1.5 will be a report documenting the sensitivity study. Subtask 1.5.3 has been assigned for the preparation of this report. At this point, the fire model can begin to be used to perform comparison studies of interest for internal fires. Of course, this will be limited by the extent to which material burning models for aircraft materials have been developed and implemented into the model.

3.6 Task 1.6: Uncertainty Analysis and Reduction

Once the fire model sensitivities have been determined, steps will be taken to more accurately characterize and reduce the uncertainty in the highest sensitivity submodels, input parameters, and environmental conditions (those which most greatly impact the end result). This will undoubtedly require the development of advanced diagnostics and measurement techniques in order to obtain data necessary to accurately determine and subsequently reduce the uncertainty associated with submodel results and input parameters. The current state-of-the-art in diagnostics and measurement techniques for fires is primitive relative to those for other combustion processes, and has resulted in a relatively poor understanding of the physics involved in a fire. The advanced diagnostics and measurement techniques will then be used to acquire the necessary data such that improved submodels (or descriptions) of the physical processes, input parameters, and environmental conditions can be developed which reduce the uncertainty associated with them.

The following subtasks have been identified:

- Subtask 1.6.1 Assess What Data is Needed
- Subtask 1.6.2 Develop Advanced Diagnostics and Measurement Techniques
- Subtask 1.6.3 Acquire Data to Characterize Uncertainty
- Subtask 1.6.4 Develop Improved Submodels and Estimates of Input Parameters and Environmental Conditions to Reduce Uncertainty

It should be noted that Task 1.3 and Task 1.5 are inter-connected with Task 1.6. As a result of the coupled physical phenomena that comprise a fire environment, changing one submodel could greatly influence the sensitivity of the end result to the output of the other submodels. Therefore, any modifications to submodels should be followed by an evaluation of the extent to which the modification has influenced the model sensitivities.

3.7 Task 1.7: Development of Additional Model Capabilities

The fourth major task of Phase 1 will involve the development of additional capabilities for the fire model. It is anticipated that some additional capabilities will be needed to model all of the fire scenarios of interest, including:

- Subtask 1.7.1 Development of Fuselage Burn-Through Submodel
- Subtask 1.7.2 Development of Aircraft Material Burning Submodels
- Subtask 1.7.3 Development of Aircraft Suppression Submodels
- Subtask 1.7.4 Development of Other Submodels of Interest

Fuselage burn-through models compatible with the types of analysis tools described herein will be developed specifically for this program in Subtask 1.7.1. These burn-through models could be very complex because of the 3-dimensional, multi-layered geometry involved, and the need to track the movement of molten aluminum.

For the material burning and suppression submodel development (Subtasks 1.7.2 and 1.7.3), the following direction is suggested:

- 1) Understand the dominant physical processes through a coupled experimental/analytical program,
- 2) Develop models of the dominant physical processes,
- 3) Incorporate submodels into research code, simulators, and RACFM.

At present, it appears that the material burning models will be developed as part of the Materials Thrust. Similarly, the suppression models may be developed as part of the Suppression Thrust. Even if these submodels are developed in these other thrusts, the physical processes are coupled to the fire environment, and effort will be required to interface them with the research code, simulators, and RACFM.

Other submodels that must be developed (including material thermal response submodels for aircraft materials) will be developed under Subtask 1.7.4.

3.8 Task 1.8: Final Validation of Research Code

Finally, to conclude Phase 1, a model validation exercise will be performed (Task 1.8). Since the application of the model will be to full-scale aircraft fire scenarios, the validation must be against large-scale data in order to include scaling effects. Every effort will be made to utilize existing large- and full-scale data for the model validation, in addition to laboratory-scale phenomenological experiments. However, it is anticipated that additional tests will be necessary to acquire the detailed data appropriate for model validation. Previous tests may not have been conducted in a manner appropriate for model validation. The performance of large- and full-scale tests to acquire appropriate validation data will most likely require additional funding resources. A multi-agency program may be necessary to generate the required resources for such an effort. Note that both DOE and DOD are currently conducting large-scale fire tests to support nuclear weapon safety assessments.

The result of Phase 1 activities will be a fully-validated fire research code for aircraft fires, and will be a major milestone in the history of fire modeling science. It will require leveraging resources and synergistic capability development in other agencies of the federal government. Interagency collaboration and cooperation will be essential to meet this goal.

4.0 Phase 2: Development of Simulator Tools and RACFM

Once the research code has been developed and validated, streamlined versions (simulator tools) and RACFM will be developed for application to specific classes of problems representative of aircraft fires. These represent the 2 major tasks associated with Phase 2:

- Task 2.1 Development of Simulator Tools
- Task 2.2 Development of RACFM

Note that it is advisable for RACFM development to begin within the first 3 years of the program, in order to assist in the integration of all the thrust areas into the vulnerability thrust.

4.1 Task 2.1: Development of Simulator Tools

As discussed previously, simulators are streamlined versions of the research code that are intended for application to well-defined classes of aircraft fire problems. There are currently 4 simulators that are envisioned for development under Task 2.1:

- Subtask 2.1.1 Simulator for Post-crash External Fires
- Subtask 2.1.2 Simulator for In-Flight Internal Fires
- Subtask 2.1.3 Simulator for Post-Crash Internal Fires (once external fire has penetrated the fuselage)
- Subtask 2.1.4 Simulator for Coupled Post-Crash External and Internal Fires

All simulators will be complete analysis packages that incorporate extensive user interfaces and pre- and post-processing capabilities.

4.2 Task 2.2: Development of RACFM

Along with the development of simulator tools, it will also be necessary to develop RACFM for the vulnerability analysis. There are many possible avenues that could be pursued for the development of RACFM: neural networks, empirical models, streamlined physics models, hybrid models, etc. Based on previous experience with the development of RACFM, the following approach is recommended.

Starting with the sensitivity analysis for the research code development, the dominant physical processes for the particular problems of interest can be identified. RACFM which explicitly model the dominant physical processes can then be developed, and the secondary physical processes can be added into the model empirically. For internal fires, this presupposes that a material burning model has been developed and incorporated into the research code. Note that the development of RACFM requires capabilities in addition the research code (e.g., neural networks, empirical data, etc.) Four RACFM will be developed to complement the four simulators developed under Task 2.1, according to the following subtasks:

- Subtask 2.2.1 RACFM for Post-crash External Fires
- Subtask 2.2.2 RACFM for In-Flight Internal Fires
- Subtask 2.2.3 RACFM for Post-Crash Internal Fires (once external fire has penetrated fuselage)
- Subtask 2.2.4 RACFM for Coupled Post-Crash External and Internal Fires
- Subtask 2.2.5 Data for RACFM Development (as needed)

The output of Phase 2 activities will be analytical tools for fire modeling of aircraft fires. They will be in a form which can be used by FAA and industry to address important fire safety issues for aircraft design and regulation. The deliverables will include simulator tools and RACFM for each class of problems of interest.

5.0 Phase 3: Continued Support and Improvement of Simulator Tools and RACFM

It is anticipated that there will be a need for continuing improvement and updating of the fire models as the state-of-the-art continues to develop. Additionally, the FAA and the aircraft designers/manufacturers who will use the simulator tools and RACFM will need to be trained to operate them properly, and to properly interpret the results. Also, technical support will be provided to help answer questions and difficulties that arise with future use of the simulators.

6.0 Milestones for Fire Modeling Plan

Detailed time estimates for completion of the tasks outlined in this plan will not be discussed due to funding uncertainty. However, time estimates will be given for major milestones in the program. The intent is to provide a vision of where the program should strive to be in future years. Reaching these milestones will require a consistent funding level (estimated at $1M - 3M per year) for the Fire Modeling Thrust. Lower funding levels will result in substantially longer completion times for these milestones. **These longer times will make it very difficult (if not impossible) to integrate the fire modeling effort with the Materials, Risk/Vulnerability, and Suppression Thrust Areas**. As shown previously in Figure 1, the Fire Modeling Thrust integrates the Materials and Suppression Thrusts into the Vulnerability Thrust. Also, close coupling of the fire modeling and materials research is necessary to ensure compatibility with each other, and the correct representation of the physical processes.

A suggested time schedule for the major milestones is shown in Figure 4. The first major milestone, the selection of 1 or 2 models for further development (Subtask 1.1.4), should be completed soon after the end of the first year. Once the model selection has occurred, development and improvement can begin.

By the end of the third year of the program, the model sensitivity study for pool fires should be completed (Task 1.3). This will enable the model to be used for trade-off calculations regarding design modifications that could enhance survivability of a fuselage in a pool fire. By the end of the fifth year, the model sensitivity study for internal fires should be completed (Task 1.5). Once the sensitivities of the model are thoroughly understood, it can be integrated with material burning models (as they are developed) to make comparisons of the impact of different materials, ventilation, and fire scenarios on the fire damage, fire propagation, and toxic gas release. This will enable the model to be used for investigations of material characteristics and design improvements that could significantly reduce internal fire problems and improve passenger and crew survivability. Note that a rigorous understanding of the uncertainty of the model is not necessary for relative comparison calculations to be performed. At this point, the fire model can begin to contribute heavily to integrating the other thrust areas.

The characterization and reduction of model uncertainty (Task 1.6) should be completed sometime during years 8 - 10, as well as the development of most of the additional submodels (Task 1.7). One exception to this is the suppression submodel (Subtask 1.7.3), which is expected to require longer time to develop due to the relatively primitive state of suppression science. The final validation of the research code (Task 1.8) should be completed during years 10 - 12.

The development of aircraft fire simulators and RACFM (Tasks 2.1 - 2.2) should be completed during years 15-20. Note that early versions of both the simulators and the RACFM should be released much earlier in the program, to ensure that the needs of the FAA and the industry will be met. This will also ensure that a user-friendly interface is developed for the industry tools.

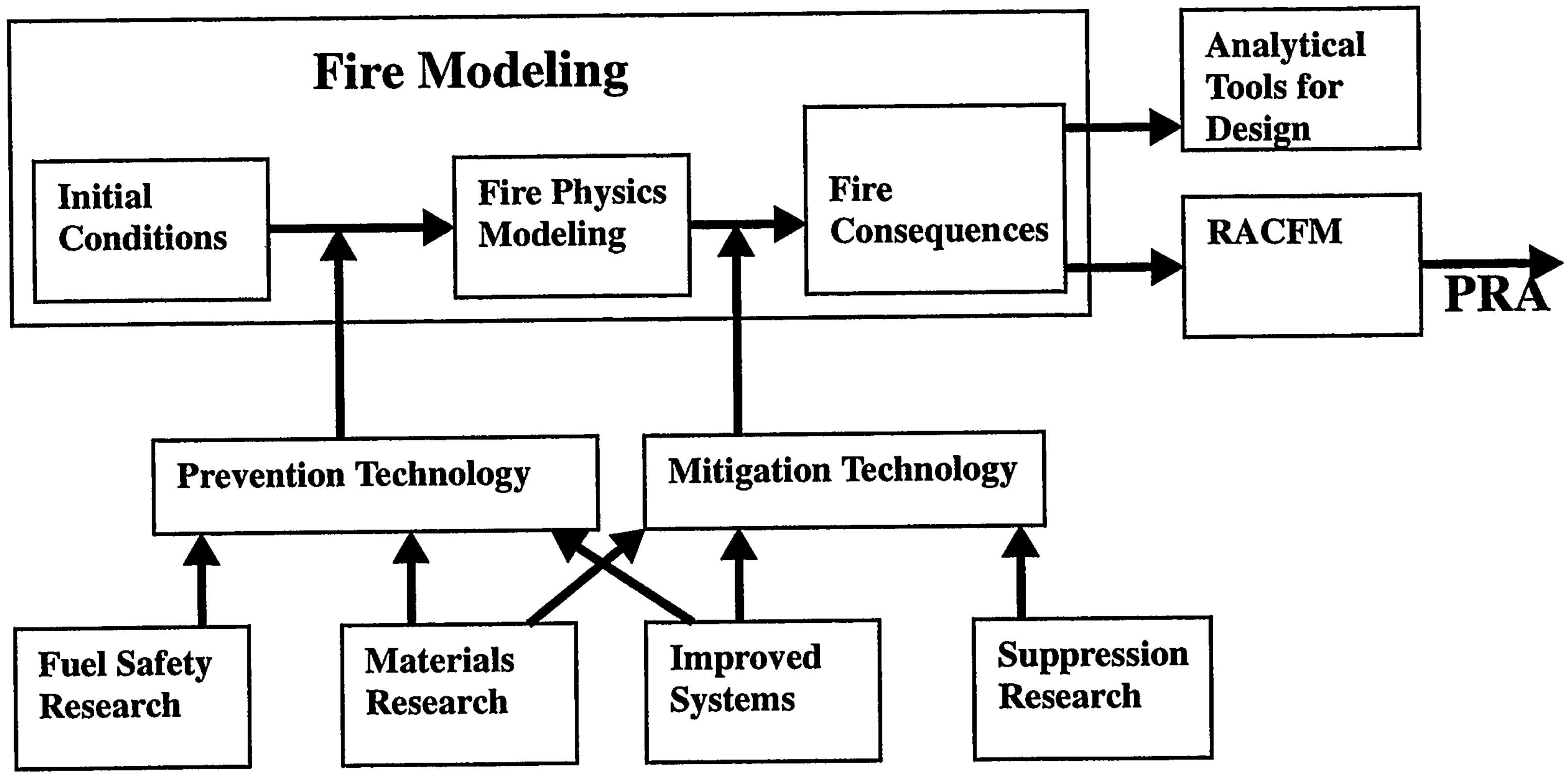


Figure 1: Fire Modeling Integration of Other Thrust Areas

PHASE 1: Development and Validation of Research Code

Task 1.1 Initiation of Fire Modeling Research

Subtask 1.1.1 Define Classes of Fire Problems of Interest for Aircraft Safety
Subtask 1.1.2 Define Technical Issues in Fire Modeling of Aircraft Fires
Subtask 1.1.3 Evaluate Existing Models for Capabilities and Development Feasibility
Subtask 1.1.4 Down-Select of Models

Task 1.2 Preliminary Validation of Model for Pool Fires

Subtask 1.2.1 Development of Numerical Thermocouple
Subtask 1.2.2 Preliminary Validation Against Large-Scale Pool Fire Data

Task 1.3 Model Sensitivity Study for Pool Fires

Subtask 1.3.1 Sensitivity to Pool Fire Input Parameters and Environmental Conditions
- Subtask 1.3.1.1 Sensitivity to Pool Fire Input Parameters
- Subtask 1.3.1.2 Sensitivity to Pool Fire Environmental Conditions

Subtask 1.3.2 Sensitivity to Pool Fire Submodel Output and Assumptions
- Subtask 1.3.2.1 Sensitivity to Thermal Radiation Submodel
  - Subtask 1.3.2.1.a Soot Submodel
  - Subtask 1.3.2.1.b Fuel Pool and Fuel Vapor Submodel
  - Subtask 1.3.2.1.c Thermal Radiation Numerical Algorithm
- Subtask 1.3.2.2 Sensitivity to Turbulence Submodel
  - Subtask 1.3.2.2.a k-ε Turbulence Modeling
  - Subtask 1.3.2.2.b Advanced Turbulence Modeling
- Subtask 1.3.2.3 Sensitivity to Numerical Solution Algorithm

Subtask 1.3.3 Documentation of Pool Fire Sensitivity Study Results

Task 1.4 Preliminary Validation of Model for Internal Fires

Subtask 1.4.1 Implementation of Existing Material Burning Models
Subtask 1.4.2 Preliminary Validation Against Large-Scale Internal Fire Data

Task 1.5 Model Sensitivity Study for Internal Fires

Subtask 1.5.1 Sensitivity to Internal Fire Input Parameters and Environmental Conditions
- Subtask 1.5.1.1 Sensitivity to Internal Fire Input Parameters
- Subtask 1.5.1.2 Sensitivity to Internal Fire Environmental Conditions

Subtask 1.5.2 Sensitivity to Internal Fire Submodel Output and Assumptions
- Subtask 1.5.2.1 Sensitivity to Material Burning Submodel
- Subtask 1.5.2.2 Sensitivity to Convective Heat Transfer Submodel
- Subtask 1.5.2.3 Sensitivity to Smoke/Species Transport Submodel
- Subtask 1.5.2.4 Sensitivity to Other Submodels

Subtask 1.5.3 Documentation of Internal Fire Sensitivity Study Results

Figure 2: Outline of DRAFT FAA Fire Modeling Plan

Task 1.6 Uncertainty Analysis and Reduction

Subtask 1.6.1 Assess What Data is Needed

Subtask 1.6.2 Develop Advanced Diagnostics and Measurement Techniques

Subtask 1.6.3 Acquire Data to Characterize Uncertainty

Subtask 1.6.4 Develop Improved Submodels and Estimates of Input Parameters and Environmental Conditions to Reduce Uncertainty

Task 1.7 Development of Additional Model Capabilities

Subtask 1.7.1 Development of Fuselage Burn-through Submodel

Subtask 1.7.2 Development of Material Burning Submodels for Aircraft Materials

Subtask 1.7.3 Development of Suppression Submodels

Subtask 1.7.4 Development of Other Submodels of Interest

Task 1.8 Final Validation of Research Code

PHASE 2: Development of Simulator Tools and RACFM

Task 2.1 Development of Simulators from Research Code

Subtask 2.1.1 Post-Crash External Fire Simulator

Subtask 2.1.2 In-Flight Internal Fire Simulator

Subtask 2.1.3 Post-Crash Internal Fire Simulator (once external fire has penetrated the fuselage)

Subtask 2.1.4 Coupled Post-Crash External and Internal Fire Simulator

Task 2.2 Development of RACFM

Subtask 2.2.1 Post-Crash External Fire RACFM

Subtask 2.2.2 In-Flight Internal Fire RACFM

Subtask 2.2.3 Post-Crash Internal Fire RACFM (once external fire has penetrated the fuselage)

Subtask 2.2.4 Coupled Post-Crash External and Internal Fire RACFM

Subtask 2.2.5 Acquire Data for RACFM

PHASE 3: Continued Support and Improvement of Simulators and RACFM

Figure 2 (Continued): Outline of DRAFT FAA Fire Modeling Plan

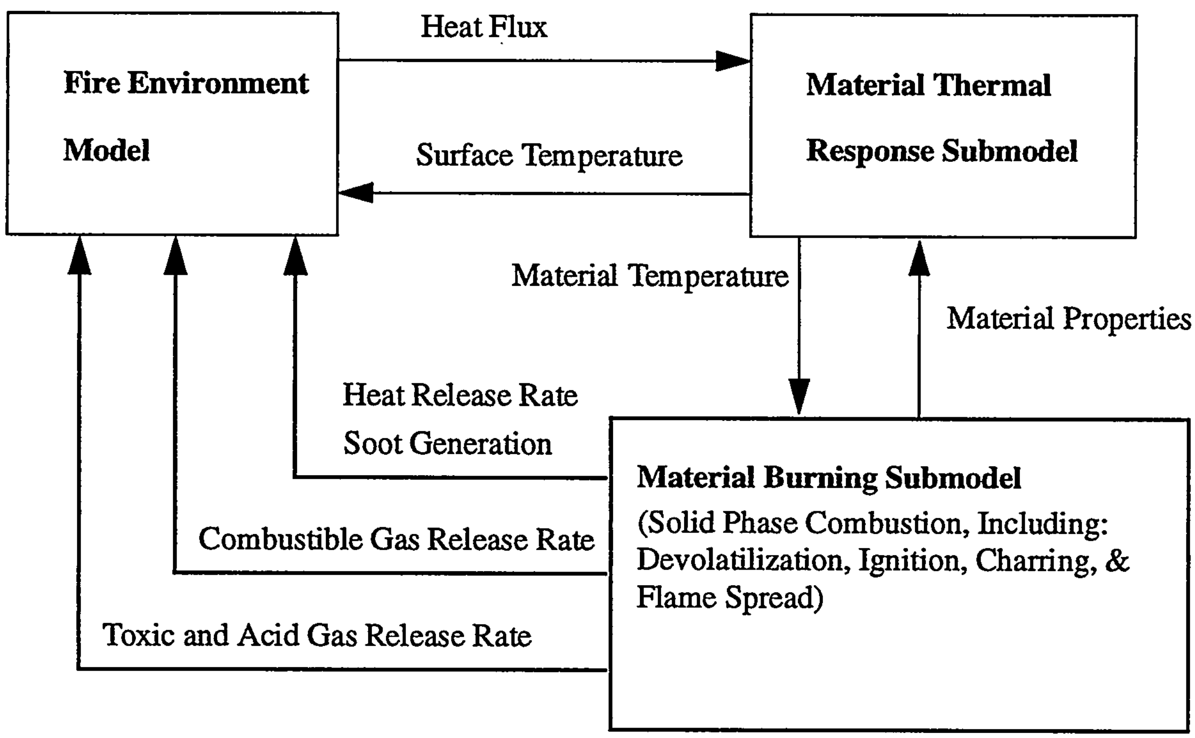


Figure 3: Interaction of Materials Models with Fire Models

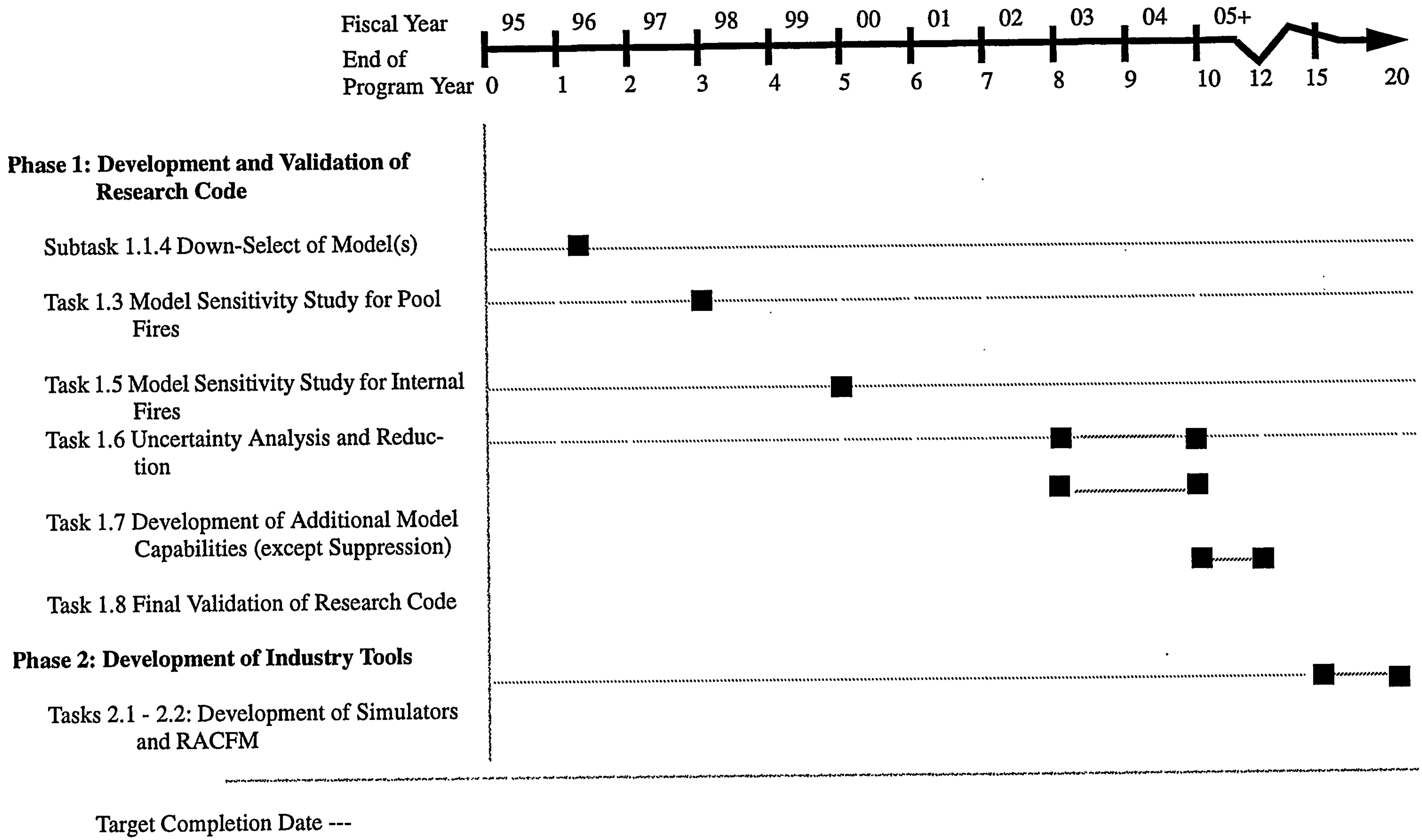


Figure 4: Deliverables Schedule for DRAFT FAA Aircraft Fire Modeling Thrust

Appendix AA: Discussion of Sensitivity and Uncertainty

In this report, 'uncertainty' is used to represent the range of possible values/outputs for a given parameter or submodel. The term 'validation' refers to comparison of a model result to experimental data. This is not to be confused with 'accuracy', which refers to comparison of a result to the truth. For real fire environments, the 'truth' is generally very difficult to determine.

To form a basis for discussion, consider a fire model with variable values at many levels, the lowest level being the input values required to run the simulation and the highest level being the end result of interest (e.g., the radiative heat flux to an object). At any level, the uncertainty associated with the end result (e.g., radiative heat flux) is given by the product of the uncertainty inherent in the variable value at that level (e.g., absorption coefficient) and the sensitivity of the end result to the variable value at that level. If all of the correct physical phenomenon are properly included in the model, the sensitivities are solely a function of physical laws. However, it is possible that a submodel has been incorrectly formulated such that it results in a non-physical sensitivity. Also, the uncertainties inherent in a variable at any level can be reduced, thereby reducing (to an extent which depends on the sensitivity) the uncertainty associated with the end result.

AA.1 Preliminary Validation

The goal of this step is to develop confidence that all of the important physical processes are represented in the model. As a result of this step, the ability of the model to predict experimental data will be quantified. The overall goal is to improve the model agreement with experimental data in the most time and cost efficient manner possible.

AA.2 Characterize Sensitivity

The goal of the sensitivity analysis is to provide information which will help direct research and development efforts such that the maximum improvement in model agreement with experimental data is obtained for a minimum expenditure of time and resources. In essence, the sensitivity analysis will determine how the different aspects of the model influence the end result. Sensitivities associated with the model input parameters, environmental conditions, and submodels will be investigated.

AA.2a Input Parameter and Environmental Conditions Sensitivities

This task represents the lowest level at which the sensitivity can be evaluated. The extent to which the model results are influenced by the input values required by the model is analyzed. Also, a knowledge of the sensitivity to environmental conditions will define the range of conditions that need to be addressed during future model development and validation. This provides a feel for the importance of accurately determining these input parameters and environmental conditions. At the conclusion of this level of the investigation, the extent to which the uncertainty associated with the input parameters and environmental conditions is propagated through the model to influence the end result will have been quantified. These sensitivities will be used to determine the level of effort which should be devoted to obtaining improved values of these parameters and conditions either by future model development or further experimental analysis.

AA.2b Submodel Sensitivities

The sensitivity of the end result to the output of the various submodels, and how those submodel results are incorporated to produce the end result, will be investigated in this task. The goal is to determine and prioritize which submodels need to be refined in order to reduce the uncertainty associated with the end result. If the end result is found to be a strong function of the output of a particular submodel, then future efforts will be devoted to improving that submodel. If, however, the results are found to be relatively insensitive to the predictions of a particular submodel, then refinement of that submodel will be assigned a lower priority. Also, the sensitivity of the end result to the use of submodel output in the overall formulation (for example, the need to increase or reduce the frequency at which submodel output is updated) will be investigated in order to determine the appropriate level of effort which should be devoted to developing an improved numerical model structure. Also, based on a sensitivity analysis of the numerical structure, it may be possible to decrease the simulation time without influencing the fidelity of the results by restricting the code to avoid updating predictions which have been shown to have a small influence on model results.

The above sensitivity study can be carried out to a number of levels. If the analysis at the lowest level shows that the submodel output significantly influences the end result, then the sensitivity analysis will proceed to the next higher (more detailed) level. Sensitivities can be investigated to determine which component of the submodel is dominating the behavior of the results. For example, if investigation of the soot submodel shows that it has a strong influence on the radiative heat flux (an important output of the model), then the sensitivity analysis will continue to the next higher (more detailed) level of examining the sensitivities associated with various aspects of the soot submodel. The soot generation, combustion, and agglomeration processes will be investigated to determine where the sensitivities are. Perhaps it is in a constant used in the soot generation algorithm, or perhaps it is an inherent assumption in the soot combustion algorithm. Once the sensitivities have been identified, research efforts can be directed toward the most sensitive areas.

AA.3 Analyze Uncertainty

After completing the sensitivity study, the extent to which low level variables (environmental conditions and empirical input parameters) and higher level variables (submodel outputs, numerical structure, submodel assumptions and inputs) influence the end result is known. In other words, the extent to which uncertainty at these levels is propagated to influence uncertainty associated with the end result is known. The goal now is to analyze (i.e. quantify) these uncertainties, starting with the variable which results in the greatest uncertainty associated with the end result (i.e., the variable value for which the highest sensitivity was identified). This will undoubtedly require the development of advanced measurement and diagnostic equipment to generate the experimental data required to quantify the uncertainty.

Since the characterization of sensitivity and the analysis of uncertainty both play a role in the uncertainty associated with the end result, it is logical to ask why the sensitivities should be characterized first. The justification for this approach is based on the cost of testing versus the cost of numerical simulations. The development of new experimental techniques and diagnostic equipment is costly. Furthermore, performing tests is often time consuming and expensive. Therefore, it is desirable to limit experimental investigations to variable values which have been identified as having a significant influence on the end results (i.e., those to which the end result is highly sensitive).

AA.4 Reduce Uncertainty

In the next step, research and development will occur to reduce the uncertainty associated with submodel outputs, input parameters, and environmental conditions. High priority will be assigned to submodel outputs, input parameters, and environmental conditions which are shown to produce a large uncertainty in the end result. For example, if a variable value has been shown to possess a large uncertainty and the end result has been shown to be highly sensitive to the variable value, then top priority would be assigned to improve the fidelity of the submodel which calculates that variable.