4th Pacific International Conference on Aerospace Science and Technology
Kaohsiung, Taiwan, 21-23 May 2001

©2001 PICAST4

Numerical Model of Flame Spread over Solids in Microgravity: A Supplementary Tool for Designing a Space Experiment

Hsin-Yi Shih and James S. T'ien

Department of Mechanical and Aerospace Engineering
Case Western Reserve University
Cleveland, Ohio 44106-7222, USA

ABSTRACT: The recently developed numerical model of concurrent-flow flame spread over thin solids has been used as a simulation tool to help the designs of a space experiment. The two-dimensional and three-dimensional, steady form of the compressible Navier-Stokes equations with chemical reactions are solved. With the coupled multi-dimensional solver of the radiative heat transfer, the model is capable of answering a number of questions regarding the experiment concept and the hardware designs. In this paper, the capabilities of the numerical model are demonstrated by providing the guidance for several experimental designing issues. The test matrix and operating conditions of the experiment are estimated through the modeling results. The three-dimensional calculations are made to simulate the flame-spreading experiment with realistic hardware configuration. The computed detailed flame structures provide the insight to the data collection. In addition, the heating load and the requirements of the product exhaust cleanup for the flow tunnel are estimated with the model. We anticipate that using this simulation tool will enable a more efficient and successful space experiment to be conducted.

INTRODUCTION: SPACE EXPERIMENT SIBAL

Flame spread over solid fuels is a classic combustion phenomenon involving the interaction among fluid dynamics, heat transfer and chemical reaction for a complex two-phase non-premixed flame. The recent modeling results [1,2] showed that flame spread and extinction phenomena in low-speed flow (less than 20 cm/s) are fundamentally different from those in higher-speed flow typically encountered on earth. Therefore, the studies of flame spreading and flammability in a microgravity environment are of scientific interest and also essential for the improvement of fire safety in spacecraft and space stations.

In a microgravity experiment called SIBAL [3] (Solid Inflammability Boundary At Low-speed), proposed to be conducted in the International Space Station, longer microgravity duration will be available to determine the flammability of thin combusting solids and the steady flame structure in low-speed, forced-concurrent flow. A novel device that facilitates the tedious process of finding the flammability boundary for a solid material has been developed and tested successfully [4]. The SIBAL experiment will validate the theoretical prediction, fill the void in experimental data and contribute to the scientific understanding of the flame at low-speed flows.

However, the limitations of space and materials, the stringent requirements of exhaust gases in the space station and the lack of opportunity to do trial-and-error testing post many challenges for the experimental design. Although not originally intended, the recently developed numerical model [2,5] for flame spreading in microgravity is being used as a supplementary tool for designing the space experiment. In the following, the essence of the model will first be described and several examples of using the model to help the experimental design will be presented.

THE NUMERICAL MODEL

Description of the model

The simulation tool includes two-dimensional and three-dimensional codes, which solves the laminar, steady, full Navier-Stokes equations for the conservation of mass, momentum, energy and species. The two-dimensional (2-D) model predicts the limiting situation and gives the qualitative trends of the flame behaviors. The three-dimensional (3-D) model can simulate the real experiment because three-dimensional effects cannot be avoided due to the limited available space of the experiment hardware in spacecraft facilities. The solid is assumed to be a thermally thin solid sheet and the solid model consists of continuity and energy equations whose solutions provide boundary conditions for the gas phase. The gas-phase reaction is represented by a one-step, second-order finite-rate Arrhenius kinetics and the solid pyrolysis is approximated by a one-step, zeroth-order decomposition obeying an Arrhenius law. The detailed mathematical formulations, thermal and transport properties can be found in [2,5].

The SIMPLER algorithm [6] is used for the fluid flow and combustion equations and the gas-phase radiation is solved using S-N discrete ordinates method [7,8], which is capable of treating multi-dimensional radiative transfer. Radiation (gaseous and/or surface) plays an important role on the microgravity flames and a key part in the model. It is responsible for the existence of the low-speed quenching limit [9] and also a heat transfer mechanism besides conduction/convection in combustion systems. These models (two-dimensional, three-dimensional codes and radiation solver) can provide some guidance for the experiment during the designing stages.

Capabilities and limitations of the model

The model can compute global flame behavior as well as the detailed flame structure. Given the primary parameters of interest (for example, flow velocity, oxygen percentage and pressure), steady flame spread rate, flame length, solid burnout rate and pyrolysis length are presented. The flame temperature and species distributions, flow field and reaction rate can also be obtained. With the coupling of the radiation solver (by discrete ordinates method), both the conduction/convection, and the radiation heat fluxes distribution can be examined. However, the gas-phase radiation calculation in three-dimension is very time consuming. For parametric studies, coupled flame-radiation computations have only been performed for the two-dimensional case. The three-dimensional model (with surface radiation but not gas radiation) has been used primarily to assess the effect of fuel width and tunnel width. But, as to be shown later, uncoupled three-dimension radiation calculation can be made to estimate the radiative heat loading to the walls of the flow tunnel. The extinction and flammability boundary can be determined theoretically.

However, there are some limitations in the present model. The model is for thin solid fuels. Since iterative procedure is used, only steady state is predicted not the transient history. For example, the model can predict the extinction limit but not the transient extinction event. The model used in this paper is for a concurrent flame spread (the primary objective of SIBAL), although a corresponding opposed-flow spread model has recently been developed [10]. Simple one-step overall gas and solid pyrolysis reactions are assumed and these reaction kinetic constants need to be calibrated. The radiation calculation is gray gas approximation with CO_2 and H_2O as participating media and their absorption coefficients are calibrated against narrow-band radiation model through a one-dimensional flame [11]. The solid is also assumed a gray surface and the radiative properties (emissivity, reflectivity) need to be determined.

SIMULATION RESULTS ON THE DESIGNING ISSUES

Test matrix and operating conditions

The determination of the extinction boundary requires performing many tests to map the burn/no burn conditions. The number of tests can grow quite large and the process can be time consuming for a space experiment in the absence of a guideline on the ranges of the operating parameters. To obtain a more efficient experiment, an initial plan of the test matrix should be in place with the help of the model. Figure 1 shows the computed flammability boundary for steady, concurrent-flow spread over a thin solid fuel (trademark Kimwipes-this is not the solid fuel to be used in the experiment. The actual fuel has been chosen only very recently. So figure 1 is used only for illustration purpose) in two-dimensional configuration. The right-hand branch is the blowoff extinction limit where flame goes out because the residence time in the flame stabilization zone is too short to complete the chemical reaction. The left-hand branch is quenching limit, which occurs in low-speed flows where flame is cooled by heat loss (principally radiation). One of the main purposes of this experiment is to pinpoint the critical low oxygen limit (the merge point of the two branches), below this fundamental limit no steady flame spread is possible for any forced velocity in this ambient environment. With this theoretical prediction, anticipated ranges of operational conditions (flow velocity and oxygen percentage, balance with nitrogen) for the experiment can be identified. For example, we are mostly interested in the quenching limits and the fundamental low oxygen limit (Point C in figure 1). So we need only have flow velocities lower than 10 cm/s.

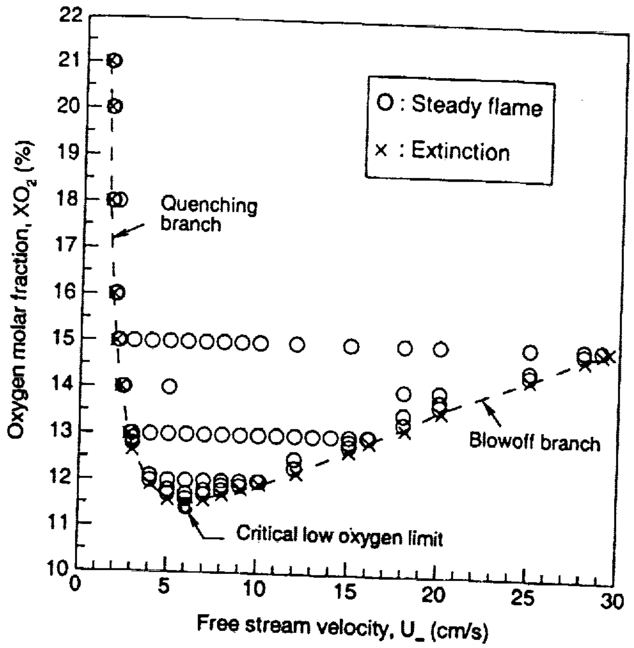

Figure 1. The flammability boundary of concurrent-flame spread over a thin solid in two-dimensional configuration.

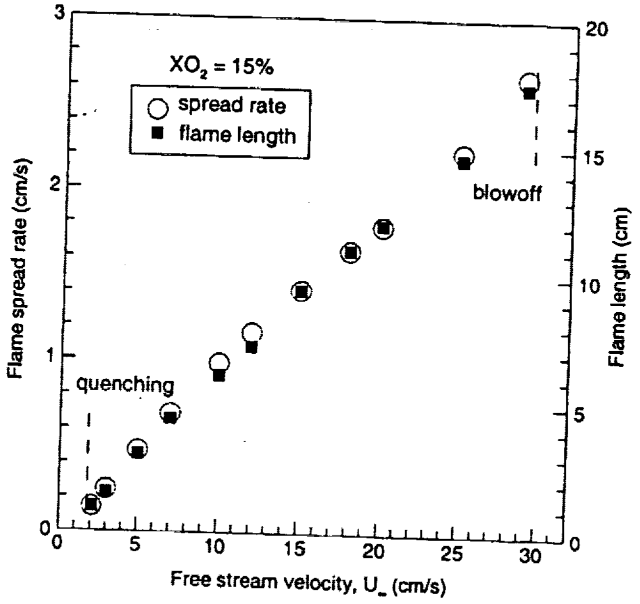

Figure 2. The computed flame spread rates and flame lengths for a traverses of flow velocity at 15% oxygen percentage.

The computed flame spread rates and flame lengths for the traverses of flow velocity at a fixed oxygen percentage (15%) in the flammability map are shown in figure 2. The steady spread rate and the flame length are approximately linear with the flow velocity. The spread rate information is used by the engineering team to design a proper solid fuel feeding device (the flame is fixed in position with respect to the flow tunnel). The flame and preheat lengths are required to determine the minimum length of the flow tunnel. Take the case with flow velocity at 10 cm/s for example. The flame length is about 7 cm. The preheat distance is of comparable length. The flame is located at 6 cm form the tunnel entrance (also determined by the model as the minimum distance without upstream conductive heat loss). This gives a total length of 20 cm. For the cases with higher oxygen percentage, the flame will be somewhat longer. So a tunnel length of 30 cm has been suggested.

Flow tunnel

The SIBAL experiment will be conducted in a small flow tunnel named FEANICS (Flow Enclosure Accommodating Novel Investigations in Combustion of Solids) inserted in the CIR (Combustion Integrated Rack) aboard International Space Station. The hardware configuration is shown in figure 3. The FEANICS tunnel is a 30-cm long, 10 × 10-cm square duct. The solid fuel is placed in the middle of the tunnel (the plane of Y = 0 cm) and therefore only upper half of the tunnel is shown in this figure. A forced-oxidizer nominal flow is imposed at the entrance and the solid fuel is feed into the flame at the necessary rate of the flame base propagation to maintain the flame stationary in the tunnel. In the experiment, the solid is fitted with a inert strip on the borders of the fuel to prevent edge burning and to facilitate the fuel feeding action. This experiment concept enables flame to have virtually unlimited time and fuel supply to reach steady state and facilitates diagnostic probing. The fuel supply device (SFDS, Solid Fuel Delivery System) to keep flame stationary in the flow tunnel has been tested in the low-gravity trial [4]. A flame is shown stabilized over the solid in the tunnel where the flame base position (X = 0 cm) is 6 cm away from the entrance. This entrance length is not finalized yet pending on the spacing of the instruments and the availability of the exposed fuel length downstream.

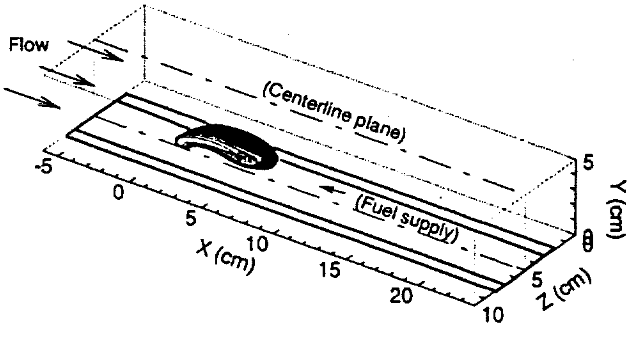

Figure 3. The flow tunnel configuration and the three-dimensional flame simulation.

In order to determine whether the cross section of the tunnel is large enough, we used the three-dimensional model to test the fuel width effect in the tunnel. Interestingly, we found [5,12] that at low flow velocity the most flammable solid sample width is not from the two-dimensional case (infinite flame width), but from a sample with an intermediate width (approximately 4 cm). This is because oxygen is limiting at low flow velocity and for the sample with this width, the flame can get extra oxygen from the sides through diffusion and at the same time is wide enough to avoid excessive heat loss. With this result, it is felt that we will test samples with widths 2, 4 and 6 cm. There is no need to go to very wide sample (and wide tunnel). So 10 × 10 cm is felt adequate. It should also be pointed out that tunnel with large cross section will require a larger supply of incoming air and more effort to clean up the exhaust.

The 3-D model also gives a higher fidelity simulation to the experiment (albeit without flame radiation). The flame image in the figure 3 is the real simulation from the model results for a 4-cm solid fuel (with additional 1-cm inert strips). Only half of the upper flame is shown due to the assumption of symmetry with respect to the centerline (Z = 5 cm). More tests need to be done in the future.

Experimental diagnostics

Selected modeling results for a 2-cm wide solid fuel at 15% oxygen and 5 cm/s inflow velocity are shown here to demonstrate the potential measurements in the experiment. The three-dimensional flame structures such as gas temperature, species distributions (fuel, O_2, CO_2, H_2O and N_2) and flow field in the FEANICS tunnel are obtained. The solid fuel burning characteristics are also predicted. Figure 4 shows the computed flame structure on the symmetry plane sliced perpendicular to the fuel centerline (Z = 5 cm) in the tunnel. Because of the symmetry with respect to the solid (Y = 0 cm), the upper half of the figure shows temperature isotherms and velocity vectors and the lower half shows the fuel reaction rate contours and velocity streamlines. Figure 5 shows the corresponding solid thickness profile (upper half) and solid temperature contours (bottom half). In this calculation, the solid is modeled with a composite of Kimwipes and 20% noncombustible material and the thermal inertia of the 1-cm inert strip is assumed 100 times of the fuel (can be changed depending on the final selection of the material). The solid thickness contours represent the unburned fuel fraction from 99% down to 0% (from right to the left) and the 0% curve indicates the burnout front where the fuel is completely consumed. The maximum temperature of the solid is located near the burnout front and the edges of the fuel are quenched by the strips.

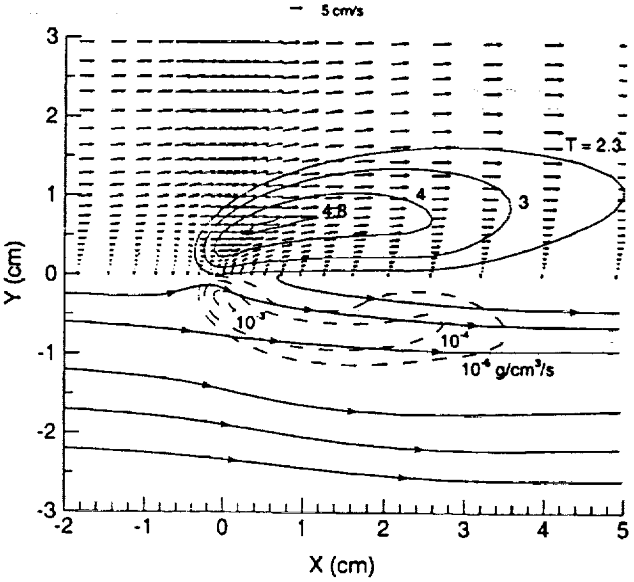

Figure 4. The detailed flame structure on the centerline plane. Upper half: velocity vectors, isotherms (1unit = 300 K), bottom half: velocity streamlines, fuel reaction rates.

These detailed flame structures (flame temperature, species distributions and solid profiles) from the model predictions will be also the representative data projected in the diagnostic measurement through the thermocouple (or thin-filament pyrometry), video, and IR cameras. The numerical model not only proves the feasibility of the experiment concept but also enable the quantitative comparisons of the experiment data.

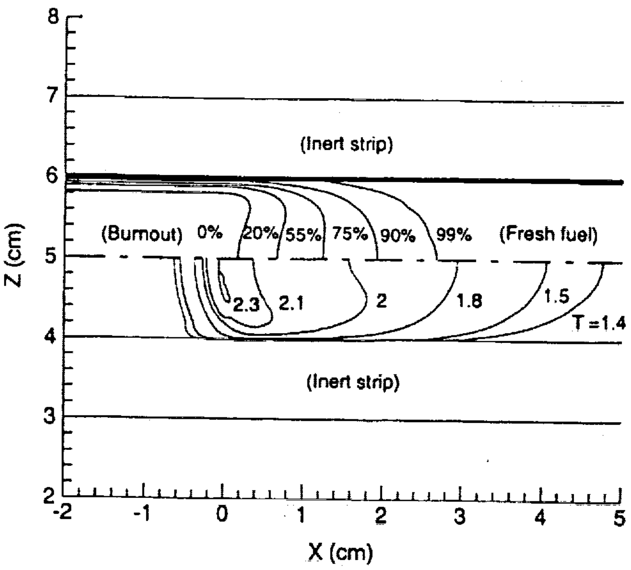

Figure 5. The solid thickness (upper half) and solid temperature profiles (lower half, 1 unit = 300 K).

Thermal management

The heat generation by the long-duration combustion experiment in space will need to be dissipated. The heat may be convected downstream and absorbed by the walls of the tunnel through convection/conduction and radiation. The heating of duct walls and/or windows may affect the diagnostic instruments and flame behavior. The heat convected downstream also needs to be cooled for the safety of the crews in space station. Therefore, the active cooling is probably required both at downstream and at the walls.

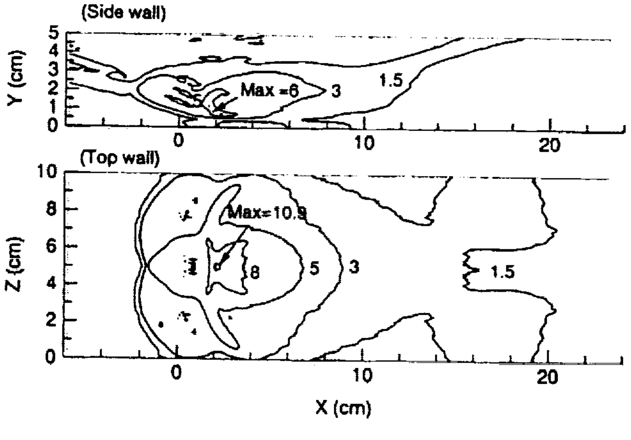

Figure 6. The incident radiation flux distribution on the tunnel wall for a 6-cm solid case (1 unit = 0.046 W/cm^2).

The wall temperature control is important for the success of the experiment. With the help of three-dimensional radiation solver, the model calculation can provide anticipated radiant heating rates so that the hardware designer in the engineering team can determine how much cooling may be needed. Here we choose the flame profile of 6-cm width solid at 15% oxygen and 5 cm/s velocity (a stronger flame and worse scenario) to perform radiation analysis. Figure 6 shows the incident radiation heat flux from the flame on the tunnel walls. The wavy distribution of the heat flux is because of the ray effect, a well-known shortcoming of discrete ordinates method due to the discretization of the angular variable [13]. From the results, the maximum incident radiant flux is 0.276 W/cm^2 on the side walls (the walls perpendicular to the solid fuel) and 0.5 W/cm^2 on the top wall (the wall parallel to the solid). The locations of large heat flux are close to the flame base region where flame and surface temperature is high (note that near the flame, the convective contribution to the wall is very small compared with that of radiation).

The overall heat loading convected downstream from the combustion system can also be estimated. We can calculate the total heat release rate due to the combustion and minus the radiation heat loss, which is the integration of radiation heat flux over the whole surface area of the tunnel walls. The balance will be the heat convected into the CIR by the flow. From the model results, the total heat release is 135.23 W where 64 W is the radiation loss and the rest of it is through convection/conduction. Figure 7 shows the temperature distribution at the exit of the tunnel. The highest gas temperature is near the middle of the duct, which is around 525 K. Along with the flow field information predicted by the model, shown in figure 8, an efficient cooling strategy can be established by the engineering team.

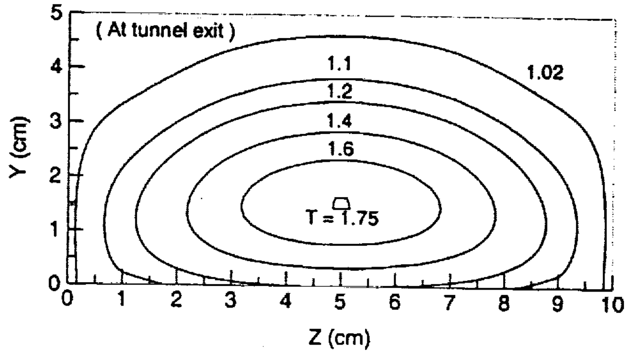

Figure 7. The temperature distributions at the exit of the tunnel (1 unit = 300 K).

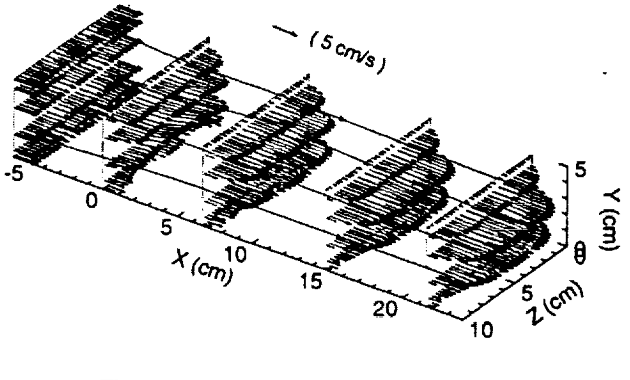

Figure 8. The flow field in the flow tunnel.

Allowable residual products in the tunnel entrance flow

For the tests with flow in the space station, a blowdown system requires a large number of gas bottles. Thus, a recirculating system is the baseline design to conserve feed gases. The exhaust products from the tunnel will go through several filters to clean up particulate, unburned fuel, CO, CO_2 and H_2O. It is then augmented with additional oxygen and is recirculated to the tunnel entrance. Depending on the efficiency of the cleanup filters, the presence of some residual products in the inlet is then possible and will affect the flame behavior and prevent the quantitative determination of the extinction boundary. The designer needs to know how much of these combustion products can be tolerated in a particular experiment. Here we use the flame spreading rate and the extinction limit as performance indicators. The model results can help to provide specifications on the humidity and residual CO, CO_2 gases concentration.

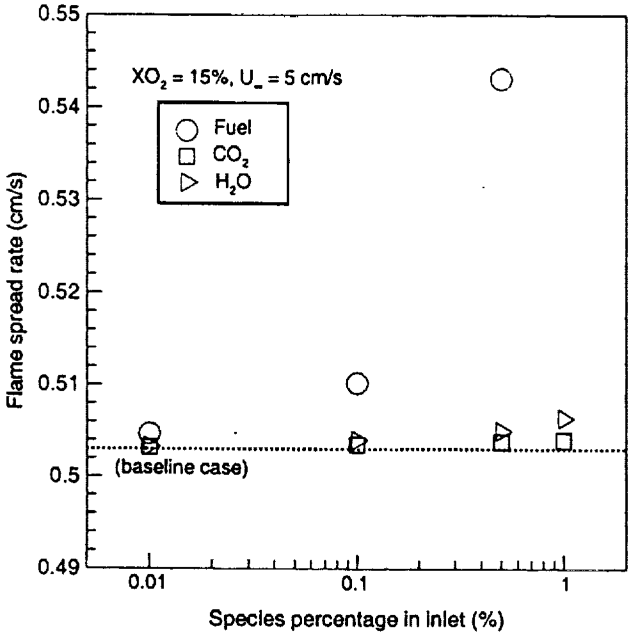

Figure 9. The flame spread rates for different residual species in the tunnel entrance flow.

The two-dimensional concurrent flame spread model is used here to determine the allowable limit of residual CO_2, H_2O species and the fuel for recirculation system (CO is simply treated as a hydrocarbon fuel in model calculations). Figure 9 shows the flame spread rates at different residual species concentrations in the inlet. The baseline case is the flame at 15% oxygen and 5 cm/s inlet velocity. The computed flame spread rate is 0.503 cm/s and the maximum flame temperature is 1553.4 K. The flame spread rate increases with increasing residual concentration. When the inlet flow includes 1% CO_2 or 1% H_2O (note that the saturation pressure of water vapor at 300 K is 0.03 atm), the flame spread rates are 0.504 cm/s and 0.506 cm/s respectively. The maximum flame temperature becomes 1544.6 K for 1% CO_2 and 1568.5 K for 1% H_2O case. The differences are so small (within 1%) that would not be influential for experiments. On the other hand, the flame is quite sensitive to the presence of residual fuel (CO or hydrocarbon fuel). When the inlet gases has only 0.1% fuel, the flame spread rate increases to 0.510 cm/s, which is 1.4% difference and the maximum flame temperature becomes 1590.6 K, which is about 2.4% difference. With 1% fuel, the flame spread rate will be even larger (0.638 cm/s).

Furthermore, the tests for different combinations these residue species are shown in figure 10. With the components of 0.1% fuel, 1% CO_2 and 1% H_2O, the flame spread rate increases is around 2%, which could be critical for some cases. However, the maximum flame temperature (1546.6 K) is close to the baseline case mainly due to the radiation loss from CO_2 emission. A test for the effect on flammability limit is also sought. With this inlet residual composition (0.1% fuel, 1% CO_2 and 1% H_2O), the low velocity extinction limit for 15% oxygen case changes from 2.1 cm/s to 2 cm/s while the low oxygen limit for 5 cm/s flow velocity case stays the same (11.7% O_2). The influence on flammability limit is not significant for this composition. To ensure the experimental concept, the allowable product gases concentrations for the recirculation system can be decided given the modeling information and the facility constraints.

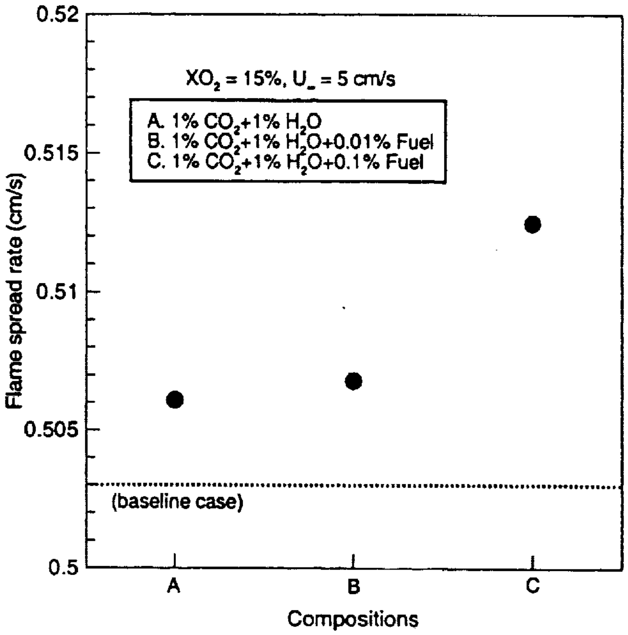

Figure 10. The flame spread rates for different inlet residual compositions.

CONCLUSIONS

Because of the unusual environment, the stringent requirements and the lack of opportunity to do trial-and error tests, space experiments must be designed more precisely than those usually conducted on earth. The examples given in this paper show how a sophisticated numerical model can be used both as a guide for the scientific design of the experiment and a supplementary tool to help resolve some of the hardware design issues.

ACKNOWLEDGEMENTS

The authors would like to thank Drs. Paul Ferkul and Kurt Sacksteder, Mr. Dave Frate and other members in the science and the engineering teams of the SIBAL project at the NASA Glenn Research Center for many helpful discussions and technical support. This work is supported by NASA Office of Biological and Physical Research through grants NCC-633.

REFERENCES

1. Ferkul, P.V. and T'ien, J.S., A model of low-speed concurrent flow flame spread over a thin solid. *Combust. Sci. Technol.*, 99, 345-370 (1994).
2. Jiang, C.B., *A Model of Flame Spread over a Thin Solid in Concurrent Flow with Flame Radiation*, Ph.D. Thesis, Case Western Reserve University, Cleveland, Ohio, USA (1995).

3. T'ien, J.S., Sacksteder, K.R. and Ferkul, P.V. *Science Requirements Document for the Solid Inflammability Boundary at Low Speed Experiment*, NASA Glenn Research Center, Cleveland, Ohio, USA (1996).
4. Ferkul, P.V., T'ien, J.S., Pettegrew, R.D. and Sacksteder, K.R., A novel device for studying flame spread over thin solid fuel. *Fall Technical Meeting of the Eastern States Section of the Combustion Institute*, Hilton Head, South Carolina, USA, #15, 171-174 (1996).
5. Shih, H.Y., *A Three-Dimensional Model of Flame Spread over a Thin Solid in Low-Speed Concurrent Flow*, Ph.D. Thesis, Case Western Reserve University, Cleveland, Ohio, USA (2000).
6. Patankar, S.V., *Numerical Heat Transfer and Fluid Flow*. New York: Hemisphere Pub. Co. (1980).
7. Fiveland, W.A., Discrete ordinates solutions of the radiative transfer equation for rectangular enclosures. *Journal of Heat Transfer*, 106, 699-706 (1984).
8. Kim, T.K. and Lee, H.S., Radiative transfer in two-dimensional anisotropic scattering media with collimated incidence. *Journal of Quantitative Spectroscopy and Radiative Transfer*, 42, 225-238 (1989).
9. T'ien, J.S., Diffusion flame extinction at small stretch rate: the mechanism of radiative loss. *Combust. & Flame*, 65, 31-34 (1986).
10. Kumar, A., Shih, H.Y. and T'ien, J.S., A comparison of extinction limits and spreading rates in opposed and concurrent spreading flames over thin solids. *2nd Joint Meeting of the Combustion Institute*, Oakland, CA, USA, March 25-28, 2001, to appear.
11. Rhatigan, J.L., Bedir, H. and T'ien, J.S., Gas-phase radiative effects on the burning and extinction of a solid fuel. *Combust. & Flame*, 112, 231-241 (1998).
12. Shih, H.Y. and T'ien, J.S., Modeling concurrent flame spread over a thin solid in a low-speed flow tunnel. *Proc. Combust. Inst.* 28, to appear (2000).
13. Chai, J.C., Lee, H.S. Patankar, S.V., Ray effect and false scattering in the discrete ordinates method. *Numerical Heat Transfer*, B, 24, 373-389 (1993).