NATIONAL TRANSPORTATION SAFETY BOARD
Investigative Hearing
Washington Metropolitan Area Transit Authority Metrorail train 302 that encountered heavy smoke in the tunnel between the L'Enfant Plaza Station and the Potomac River Bridge on January 12, 2015

GROUP	C
EXHIBIT	
9	

Agency / Organization

WMATA

Title

WMATA Outer B Route CFD Simulation Emergency Ventilation and Smoke Control Analysis
March 10, 1998

Docket ID: **DCA 15 FR 004**

WMATA OUTER B ROUTE
CFD SIMULATION

EMERGENCY VENTILATION AND SMOKE CONTROL ANALYSIS

FINAL REPORT

10 March 1998

Prepared for :
Deleuw, Cather & Company
for
Washington Metropolitan Area Transit Authority
Washington, D.C.

by
Parsons Brinckerhoff Quade & Douglas,Inc.
New York City, New York

TABLE OF CONTENTS

LIST OF FIGURES

ABSTRACT

The Outer B Route is a newly built tunnel connecting Wheaton and Glenmont stations. Three vent shafts exist between the two stations. The original ventilation shaft capacities are 200,000 cubic feet per minute (cfm) exhaust, 140,000 cfm supply. An alternative that is being designed would upgrade the capacities to 290,000 cfm supply, and 203,000 cfm exhaust. An analysis of heat and smoke conditions from a subway car fire at different locations in the route was performed to evaluate emergency ventilation system requirements and operating modes to maintain tenable conditions for evacuation in the event of a train fire occurring in this tunnel. The analysis was done using a fully developed fire or post-flashover fire. At this point of burning the heat release is at its maximum and all the combustible items in the compartment are involved and flames appear to fill the entire volume.

The analysis employed the use of FLUENT, a general-purpose computational fluid dynamics (CFD) computer program, to model the relevant physical phenomena. These phenomena include the effect of buoyancy on smoke, heat distribution, and effect of mechanical ventilation on smoke movement.

The analysis examined heat and smoke migration patterns caused by 42.4 million Btu per hour (MBtu/hr) fire. The environmental conditions were compared to the tenable environment criteria recommended in the National Fire Protection Association's Standard 130, "Fixed Guideway Transit Systems), (NFPA-130), 1997 edition. They include a temperature of 140°F or less and a light extinction coefficient of 0.2 m^{-1} or less.

1. INTRODUCTION

Fire-life safety is a major concern in subway stations and tunnels. During subway emergencies involving fire or the generation of hot gases, the products of combustion can be toxic, reduce visibility and have high temperatures, all of which can endanger subway patrons and fire-fighting personnel. The control of smoke movement and the evacuation of passengers work hand-in-hand. Ventilation is of primary importance in controlling the direction of spread of heat and smoke, and providing passengers with a relatively safe evacuation path. The prediction and control of heat and smoke movement within stations and tunnels is necessary for formulating an emergency response plan, and designing strategies to provide for the safe evacuation of passengers.

The Authority's evacuation policy is stated to be unidirectional. That is, for each section of tunnel, there is a pre-determined direction of evacuation, regardless of where the fire is on the train. For tunnel sections having grade, the direction of evacuation is always downhill.

2. METHODOLOGY

During recent years, Computational Fluid Dynamics (CFD) has made it possible to obtain numerical predictions of complex situations involving fluid flow, heat transfer, and mass transport. These computations give useful design information and provide interesting insights into the physical processes. Computational analysis enriches experimental information by supplying the details that may be difficult or impossible to measure.

With the continuing cost reduction and increased speed of computers, CFD is moving from the research labs of highly trained academics into the mainstream of everyday engineering. Especially important is the emerging power of personal computers that allow routine analysis on machines that are usually dedicated to a single person, and therefore are convenient to use.

There are many practical advantages of performing a computational analysis of a physical situation. It can usually be done at much lower cost than what it is required for the corresponding experimental testing. The computational investigation takes significantly less time. Whereas only few overall quantities can be conveniently measured in a normal experimental study, the computational solution gives complete details of the distributions of velocity components, pressure, temperatures, and species mass concentration.

As a result, the computational predictions not only give useful quantitative information for design but also provide valuable insight into the underlying physical processes. Whereas it is important to validate the computational results by comparison with representative experimental data, the computational analysis can

now be used to supplement and enrich an experimental investigation. This is accomplished by reducing the amount of experimental testing through the optimization of a particular design.

The aim of CFD calculations is to predict the distributions of velocity, pressure, temperature, concentration, and other relevant variables throughout the domain of interest.

The prediction of fluid flow, heat transfer, and other related processes involves numerical solution of the differential equations that describe the laws governing these processes. These equations include the conservation equations of mass, momentum, energy, chemical species. The solution procedure used in FLUENT is based on the control volume method (also known as the finite-volume method).

The steps involved in obtaining a numerical solution are as follows: the computational domain is first divided into a number of non-overlapping control volumes and a grid point is placed at the center of each control volume. The lines joining the grid points are called grid lines. The value of an internal grid point is assumed to prevail over the control volume around it. Similarly the value of a boundary grid point is assumed to prevail over the face of the associated control volume.

The discretization procedure requires an assumption regarding the variation of the dependent variable between the grid points. In this project the power-law differencing scheme is used. The coupling between the velocity and the pressure fields is handled using Semi-Implicit Method for Pressure-Linked Equation known as the (SIMPLE) algorithm. For a time-dependent calculation, the FLUENT package uses the implicit scheme for the discretization of the unsteady terms in the equations.

The purpose of CFD simulation is to determine the value of velocity, temperature, chemical species concentration, and any other variable of interest at every grid point in the computational domain. To obtain these values, the relevant differential equations are transformed into algebraic equations by integration over each control volume. These algebraic equations contains as unknowns the values of the dependent variables at the various grid points in the domain. These unknowns are obtained by solving this set of algebraic equations. CFD simulations involve two types of variables: vector and scalar. A scalar variable is one that has a value at a point but no direction. For example, temperature is a scalar variable because it has a value at a given point, but no direction. A vector variable has both a magnitude and a direction. Because velocity is a vector variable, its magnitude and direction are determined by solving for the magnitude of its velocity components in each of the coordinate directions.

2.1 FIRE REPRESENTATION

In the present model, fire is represented as a source of heat and mass. The model does not simulate the combustion process. Instead, the heat release rate due to combustion is prescribed as a volumetric heat source. The heat release rate and the heating value of the fuel are used to compute the mass flow rate of the fuel consumed. To simulate the formation of products during the combustion process, an extra equation is solved for the products of combustion. The local concentration of smoke will be related to the concentration of the products using the yield rate of smoke of the fuel used. The conservation equation of the combustion products contains a source term which is deduced from the rate of the fuel consumption and the stoichiometric ratio (air to fuel mass flow rate ratio for a complete combustion). It should be noted that, due to the differences in the boundary conditions at the tunnel walls and the nature of the source terms in both the energy conservation equation and the species conservation equation, the distributions of temperature and the products of combustion profiles, in general, will be different.

2.2 COMPUTER SIMULATIONS

The section of the tunnel simulated was 897 ft long with a semi-circular cross section with a radius of 7.5 ft as seen in figures 1a and 1b. The train consists of eight cars, each car is 74.75 feet. The fire location was always chosen in the sixth car in the direction of airflow. Conduction of heat in the train walls as well as heat loss through the tunnel walls were taken into account. Eleven simulations were performed. The following table will describe the nature and boundary conditions of each run :

Scenario	Simulation	Mode	Tunnel Airflow Rate (kcfm)	Supply Grade (% direction)	Nomenclature
1	1	Transient	39.80	0.35 Up	1-A-TR-SM2-FI
1	2	Steady	39.80	0.35 Up	2-A-SS-SM2-FI
1	3	Steady	39.80	0.35 Up	3-A-SS-SM2-PI
1	4	Steady	39.80	0.35 Up	4-A-SS-SM3-FI
2	5	Steady	3.043	4.00 Down	5-B-SS-SM3-FI
3	6	Steady	51.96	0.35 Up	6-C-SS-SM3-FI
4	7	Steady	16.232	4.00 Down	7-D-SS-SM3-FI
4	8	Steady	25.00	4.00 Down	8-D-SS-SM3-FI
4	9	Steady	35.00	4.00 Down	9-D-SS-SM3-FI
4	10	Steady	40.00	4.00 Down	10-D-SS-SM3-FI
4	11	Steady	45.00	4.00 Down	11-D-SS-SM3-FI

In the nomenclature, the first character is the order the cases were analyzed. The second character, a letter, refers to the scenario case. The next two letters indicate if the case is steady (SS) or transient (TR). The fourth indicator shows which of the PB heat and smoke generation models (SM1, SM2 or SM3) was used. The fifth term indicate if the fire was fully distributed among the car on fire interior (FI) or partially distributed through the car interior (PI).

The first scenario was run four times in order to decide about the adequacy of the final fire model to be used. The aim of the first and the second run was to see the differences that will result in our conclusions by running the case transient or steady. The fire was spread evenly in the entire car for both cases. Next case, the fire was chosen to be distributed in a smaller volume of the car. The fourth case that was run using the first scenario was intended to test the revised smoke model. The differences between the newly implemented model (SM3) and the one that was used before lie in the user subroutines that were attached to the main program to simulate the heat and the smoke source terms that was caused by the existence of the fire in the computational control volume. After this initial step of testing, the rest of scenarios were run. Each with different supply airflow as can be seen in the table.

For boundary conditions the tunnel inlet cross section was used as velocity inlet, while the other end was used as a pressure boundary. The ambient temperature was chosen as 95°F, which represent summer conditions in Washington, D.C. The walls of the train and the tunnel conductance of heat was included in the analysis. Variation of the physical properties with temperature, pressure and contamination concentration was considered and embedded in the program.

Turbulence is a major phenomena in fires. To simulate turbulence many options are available to the user in FLUENT software. In the present study, the two equations standard k-e model was used. The effect of buoyancy on the generation of turbulent kinetic energy and its dissipation was included.

3. DESIGN CRITERIA

The 1997 edition of NFPA-130 is the industry standard for fire protection and life safety. The principals and objectives of this standard were applied in this study. Design objectives of the emergency system are as specified in NFPA 130, Section 4-2.1, a-c:

- To provide a stream of non contaminated air to passengers in a path of egress away from a train fire,
- To produce rates to prevent backlayering of smoke in a path of egress away from a train fire, and
- To limit the air temperature in a path of egress away from a train fire to 140 F or less.

For the purpose of analyzing the results of simulations of various ventilation scenarios for the route, the first requirement was related to the smoke obscuration as recommended in Appendix B-1 of the NFPA 130. The light extinction coefficient that is related to the visibility must be kept 0.2 m^{-1} or less.

The goal of an emergency ventilation system for tunnels is to control the direction of smoke movement in order to provide a smoke-free path for passenger evacuation. In a situation where a transit car is on fire in a tunnel, the ventilation system forces air past the burning vehicle such that smoke and hot gases are forced away from other cars, in a direction opposite the path of passenger evacuation. When the fire occurs, the smoke and hot gases rise due to buoyancy. Operation of the emergency ventilation system will set up a longitudinal airflow pattern in the tunnel. If the ventilation system is inadequate, smoke and hot gases could start moving in a direction opposite to that of the forced air, a phenomenon called "backlayering". Backlayering will cause movement of smoke in the direction of evacuating passengers, posing a serious life safety threat.

4. DISCUSSION OF RESULTS

The results of both the temperature and the attenuation contours in all the figures were plotted at two locations of the analyzed cross section. One location is along the walkway and the other is in the middle of the tunnel. These locations corresponds to the indices of K=18 and 10 respectively. These location can be identified in the figures 1b and 1c. These figures represent the cross section of the tunnel in the physical and the computational domain respectively. For the purpose of clarification, two ranges were chosen to be shown in all the contour plots. The scale on the left hand side of these contour plots indicates the ranges represented by the two colors. For both temperature and attenuation plots, the cut-off values were chosen as 140°F and 0.2 m^{-1} respectively. These cut-off number were chosen to discern easily between the regions that pass or fail the NFPA-130 requirements.

The first scenario that was dealt with, consisted of an uphill airflow of 39870 cfm at grade of 0.35 percent.

As seen, in figures 2a, 2b, 3a and 3b the contours of temperatures and attenuation coefficient at both locations showed the same conclusions vis-à-vis backlayering phenomena. Either transient or steady analysis predicted that the hot fumes are controlled at a distance of 175 feet upstream of the car on fire. On these grounds, a choice of a steady analysis was made. Running time for a steady analysis is much shorter than the transient analysis. Also the critical issue of the time step to accurately model the problem at hand while treating the problem in a transient fashion will not be a problem for steady cases.

Next the volume where the heat and smoke source exist is reduced to see if any improvement can be seen in the inside of the car on fire. By looking at figures 4a and 4b, which depict the temperature and attenuation coefficient contours for this case, no major differences was noticed. Thus, the volume of the fire source was kept as before, occupying the entire car. For the fourth simulation, the effect of smoke generation was considered in all the conservation equations. As a result, additional terms were included in all the eight conservation equations (SM3) instead of being added only in the energy equation and the smoke equation (SM2). Figures 5a and 5b show the resulting temperature and attenuation coefficient of this run. It is observed that backlayering of smoke was under control about one car length upstream of the car on fire. The second scenario which consisted of downhill airflow of 3043 cfm at a grade of 4 percent is shown in figures 6a and 6b. These figures clearly indicate that the ventilation rate used was not adequate and sufficient to control the smoke. In other hand, the next case which consisted of an uphill airflow of 51960 cfm at a grade of 0.35 percent showed that evacuation is possible and enough ventilation was provided to control the smoke and the high temperature migration. These observations are concluded from looking at figures 7a and 7b. The last scenario was a downhill supply of 16232 cfm at a grade of 4 percent. As can be seen from figures 8a and 8b, the limits set by the NFPA-130 were not achieved.

Based on this conclusion for the last run, number of runs were tried afterward to find the airflow that would prevent backlayering and provide safe evacuation of the patrons. Four different flow rates were tried, 25000 cfm, 35000 cfm, 40,000 cfm, and 45000 cfm. The contours of temperatures and attenuation coefficients are shown in Figures 9-12. Every time the flow rate was increased the conditions in the space modeled, the backlayering encountered more resistance. To control backlayering and thus provide a space that is safe for evacuation according to NFPA-130 tenability limits, 45000 cfm was sufficient.

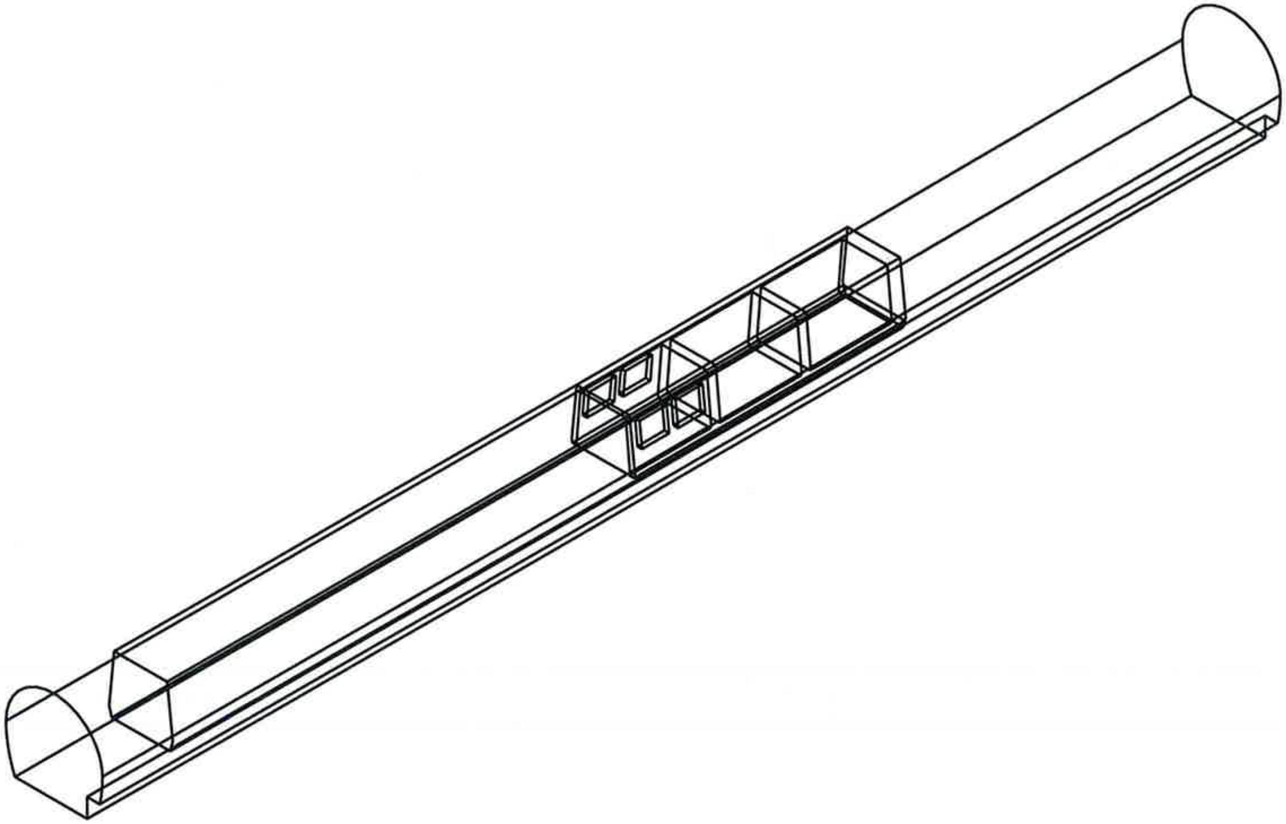
5. CONCLUSIONS

Modeling of fire Emergencies in the Outer B route tunnel section connection Wheaton and Glenmont stations was investigated using Computational Fluid Dynamics (CFD). The outcome had provided some important insights into air temperature and smoke concentration under the selected fire scenarios.

The analyses were done for two locations with two different grades using the original ventilation shaft capacities as well as the newly designed values. The grades were 0.35 percent upgrade and 4 percent downgrade. The analysis demonstrated that the fire in the 0.35 percent upgrade can be managed by either the existing values of the provided air volume flow rate or the newly designed values. Because the new alternative is 50 percent higher, the backlayering is better controlled and provide more favorable conditions for emergency evacuation. In the other hand, the results of the simulation of the second location with the 4 percent down grade show insufficient flow to prevent backlayering into the entire train-tunnel annulus. Thus the conditions, at which an emergency evacuation process is done, would not conform to the tenable environment criteria recommended by the NFPA-130. Further analyses were performed that concluded the airflow to provide the right conditions for evacuation was about 45000 cfm.

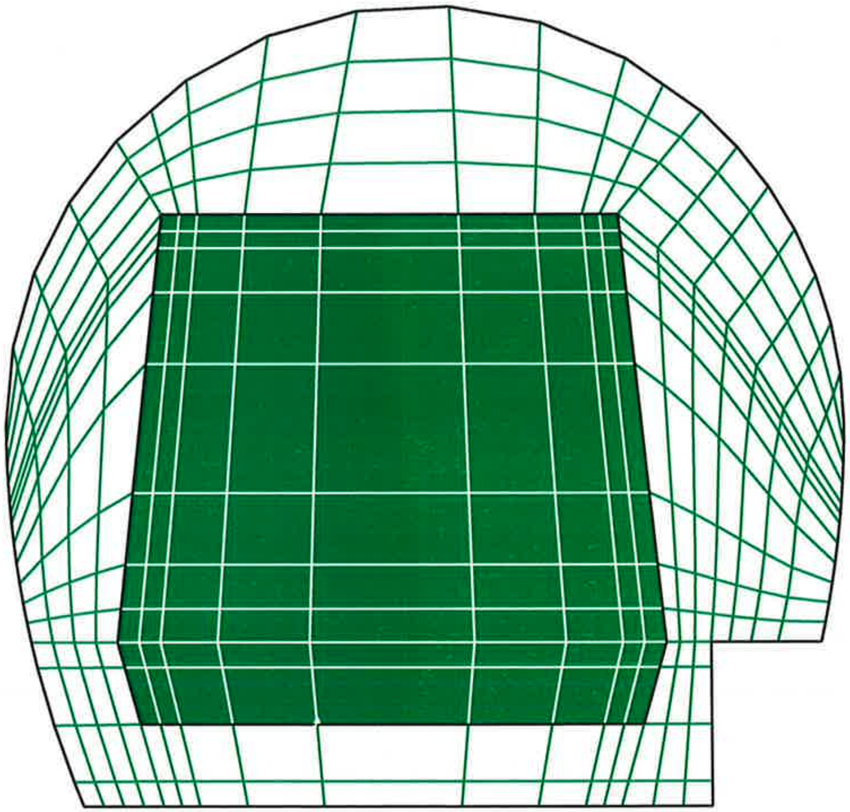
6. REFERENCES

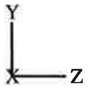
1. NFPA-130, "Standard for Fixed Guideway Transit systems", National Fire Protection Association, 1997.
2. WMATA Phase -II Tunnel Ventilation Project Report, prepared by Parsons Brinckerhoff, July 1991.
3. Drysdale, "An Introduction to Fire Dynamics", John Wiley, 1986.
4. Klote, John H and Milke, James A, "Design of Smoke Management Systems", ASHRAE/SFPE, Atlanta, 1992
5. The SFPE Handbook of Fire Protection Engineering, First Edition, National Fire Protection Association, Quincy, MA, 1988.



Configuration Outline

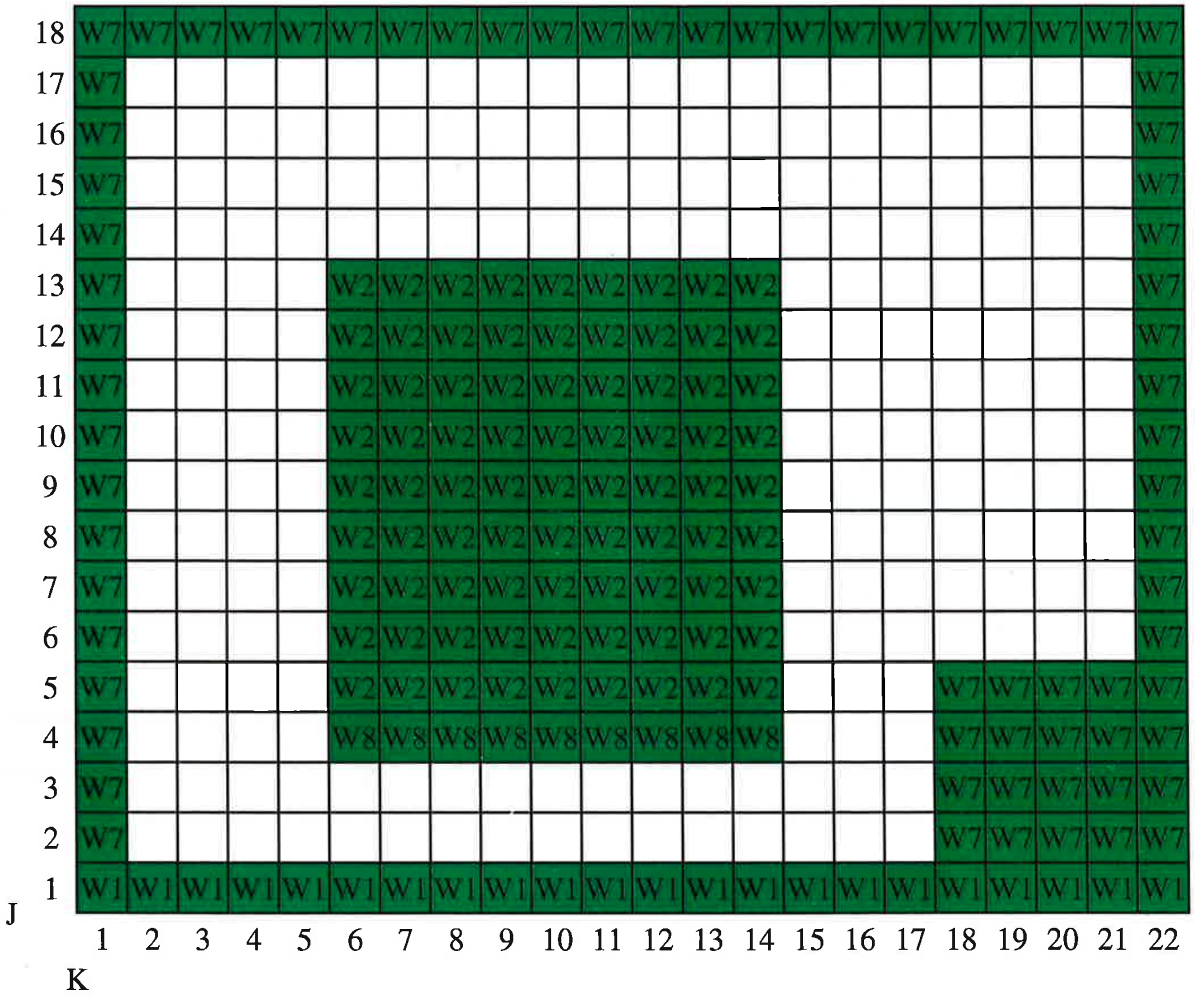
Figure 1a
Parsons
Brinckerhoff





Configuration Cross Section

Figure 1b
Parsons
Brinckerhoff

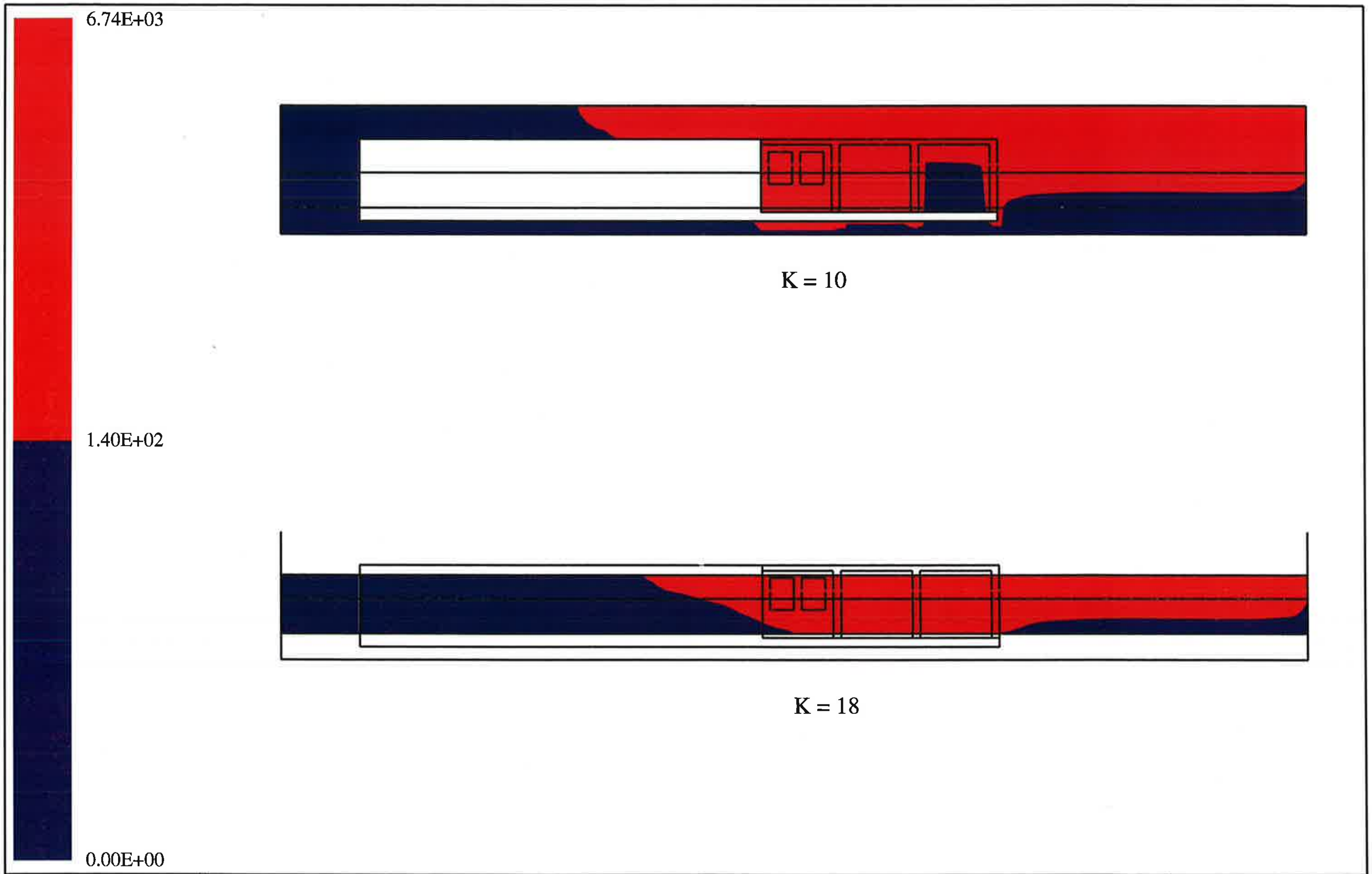


18	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7	W7
17	W7																					W7
16	W7																					W7
15	W7																					W7
14	W7																					W7
13	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2								W7
12	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2								W7
11	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2								W7
10	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2								W7
9	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2								W7
8	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2								W7
7	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2								W7
6	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2								W7
5	W7					W2	W2	W2	W2	W2	W2	W2	W2	W2				W7	W7	W7	W7	W7
4	W7					W8	W8	W8	W8	W8	W8	W8	W8	W8				W7	W7	W7	W7	W7
3	W7																	W7	W7	W7	W7	W7
2	W7																	W7	W7	W7	W7	W7
1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1	W1
J	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22

K

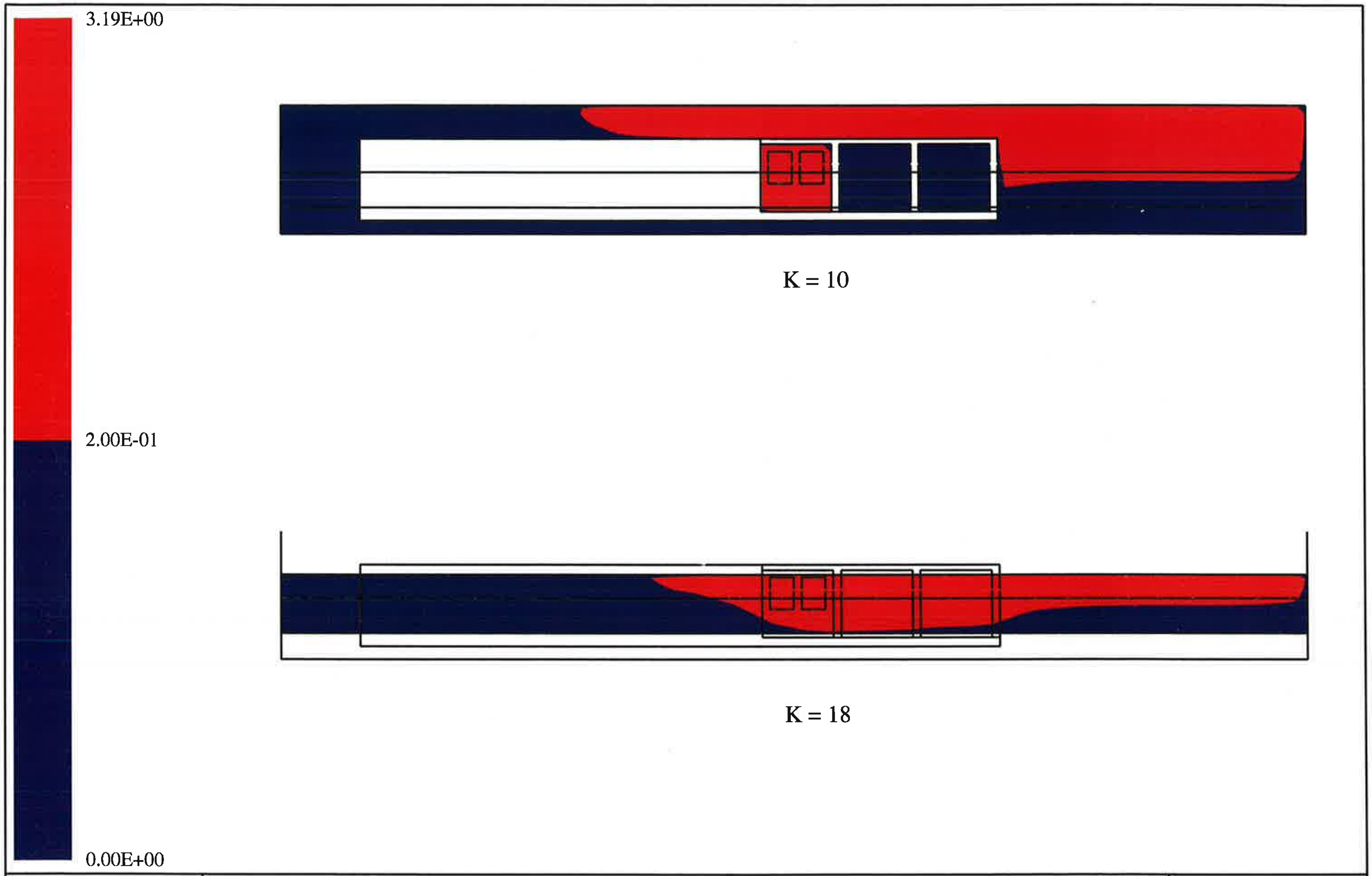
Configuration Cross Section in The Computational Domain

Figure 1c
Parsons
Brinckerhoff



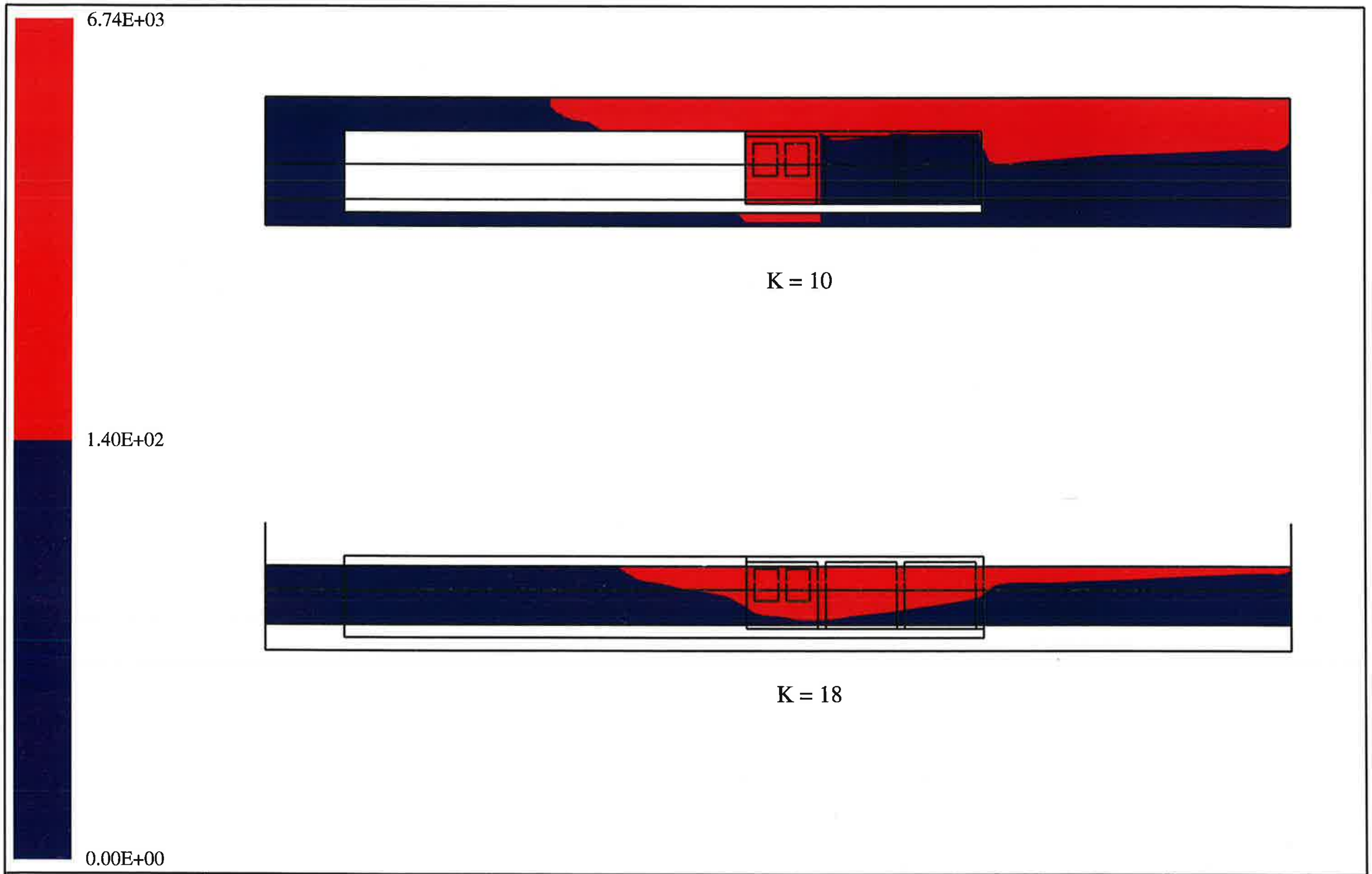
1-A-TR-SM2-FI
Temperature (Deg F)
Scenario 1

Figure 2a
Parsons
Brinckerhoff



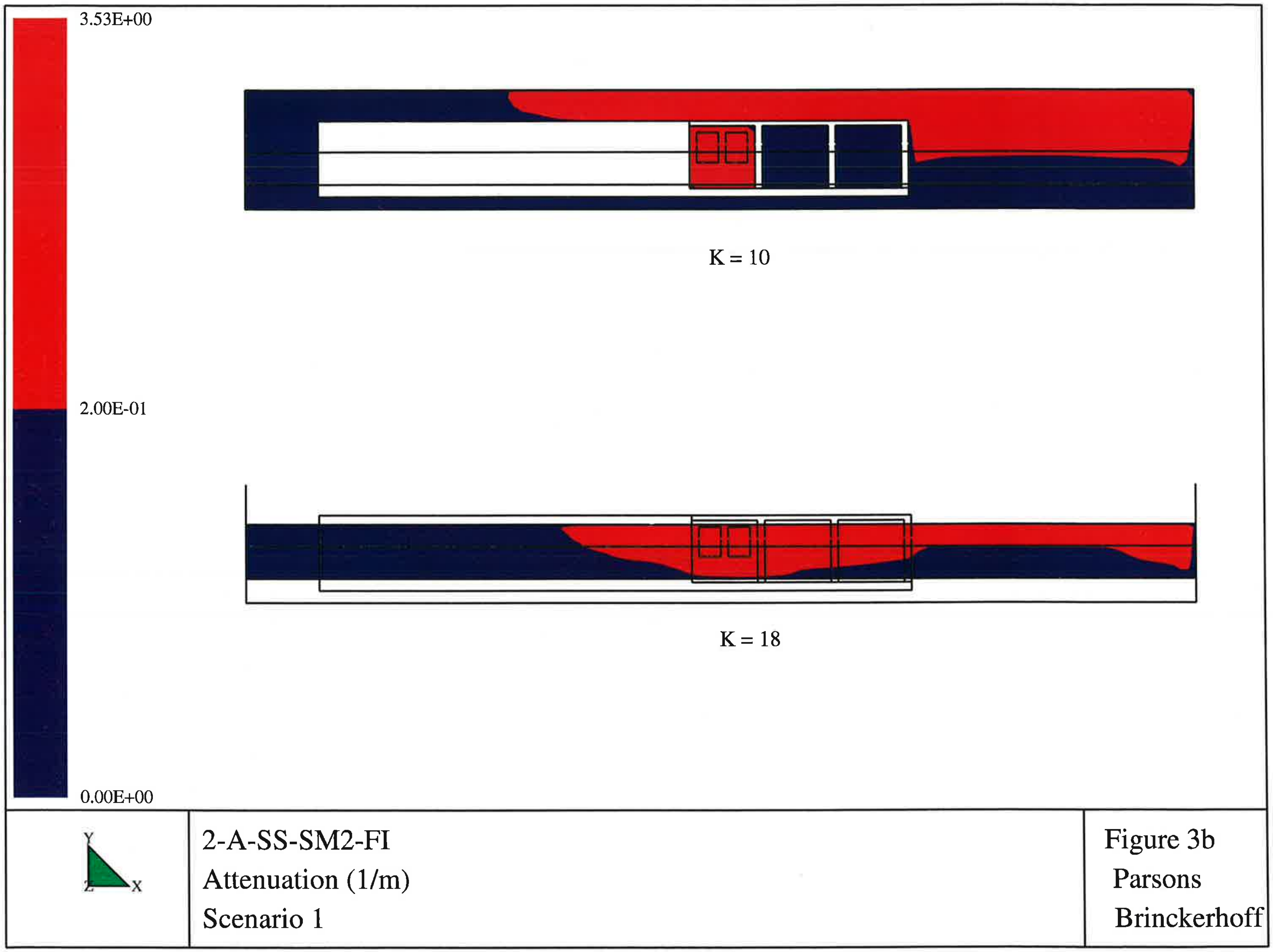
1-A-TR-SM2-FI
Attenuation (1/m)
Scenario 1

Figure 2b
Parsons
Brinckerhoff



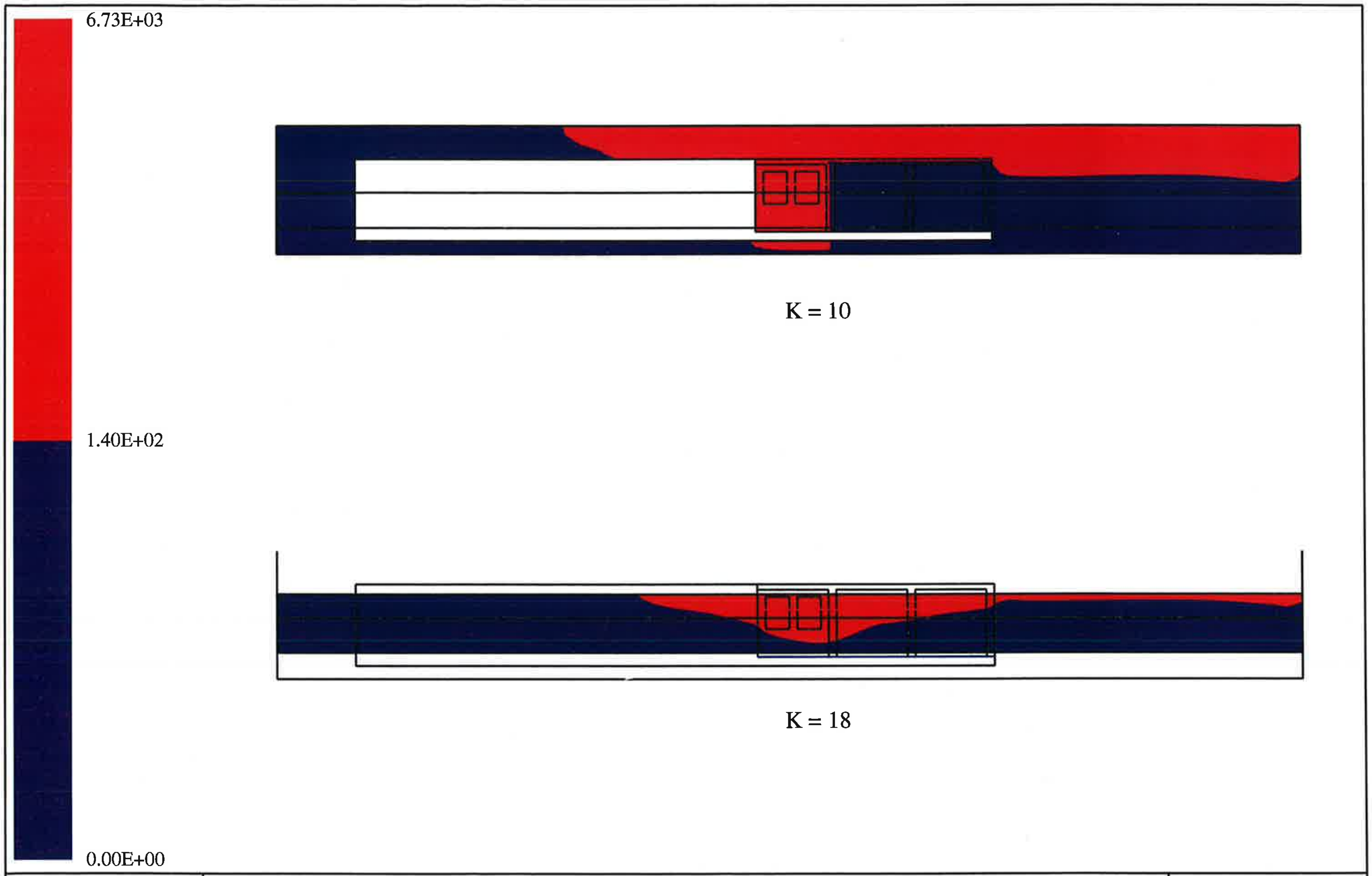
2-A-SS-SM2-FI
Temperature (Deg F)
Scenario 1

Figure 3a
Parsons
Brinckerhoff



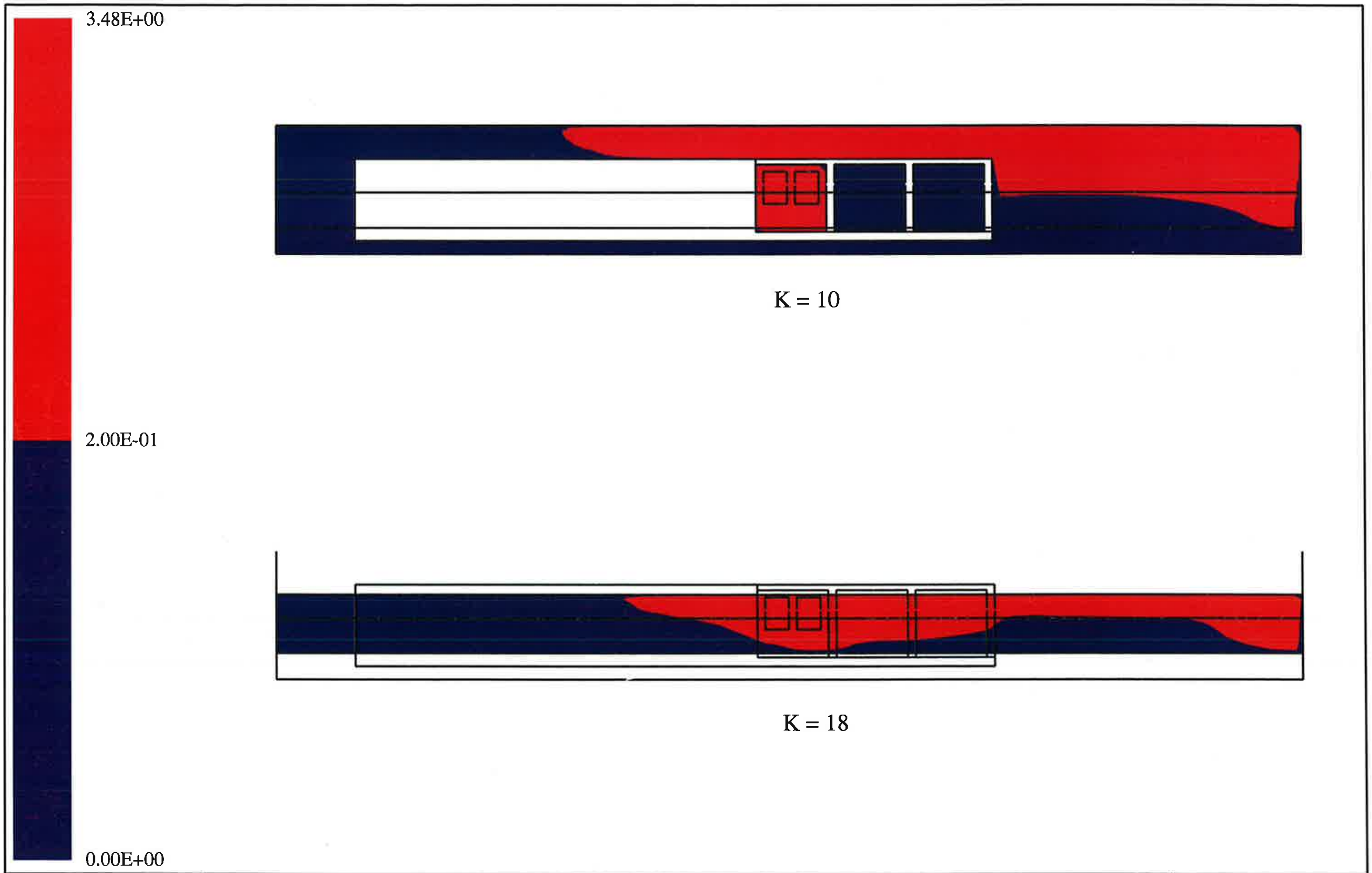
2-A-SS-SM2-FI
Attenuation (1/m)
Scenario 1

Figure 3b
Parsons
Brinckerhoff



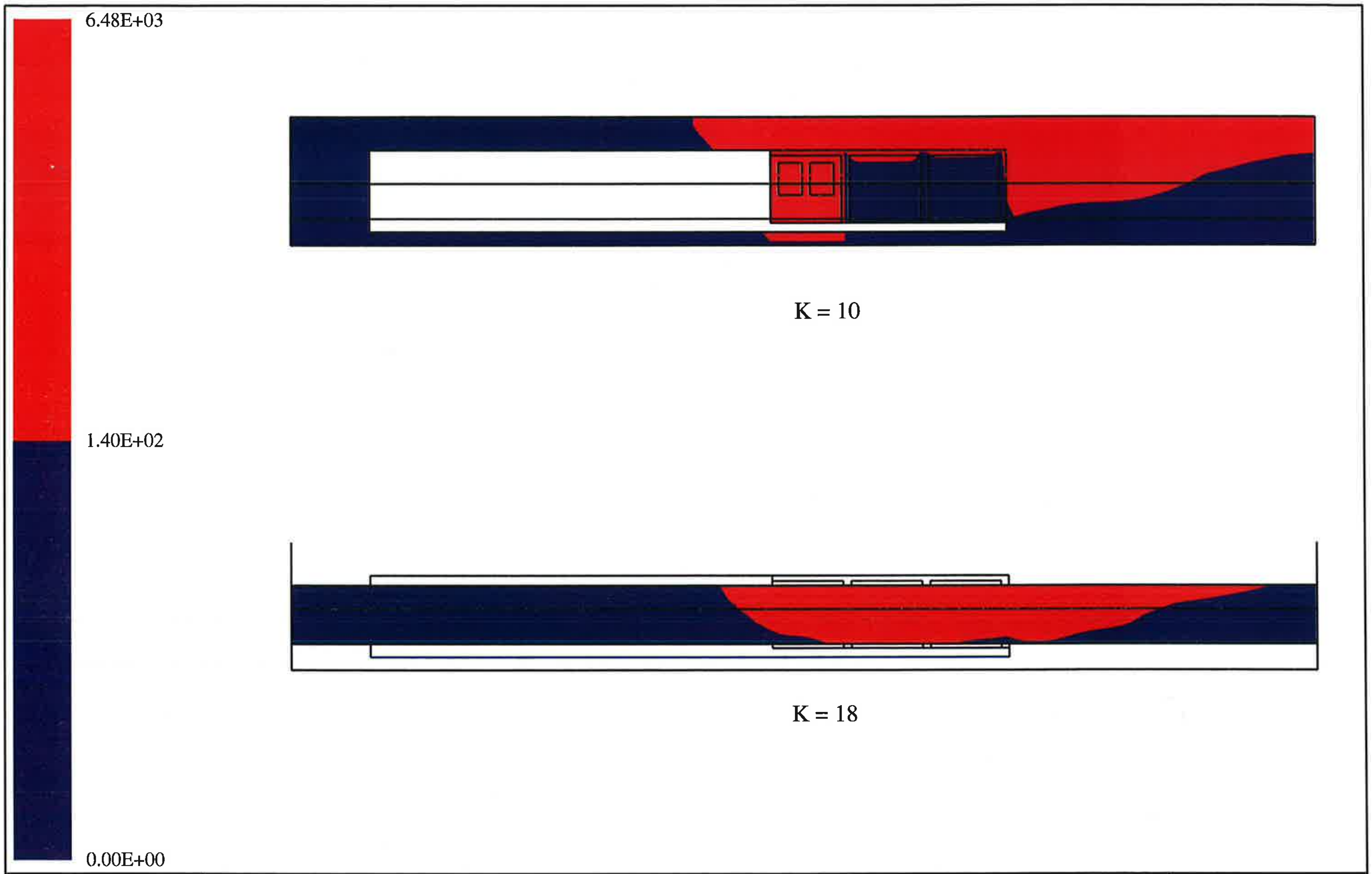
3-A-SS-SM2-PI
Temperature (Deg F)
Scenario 1

Figure 4a
Parsons
Brinckerhoff



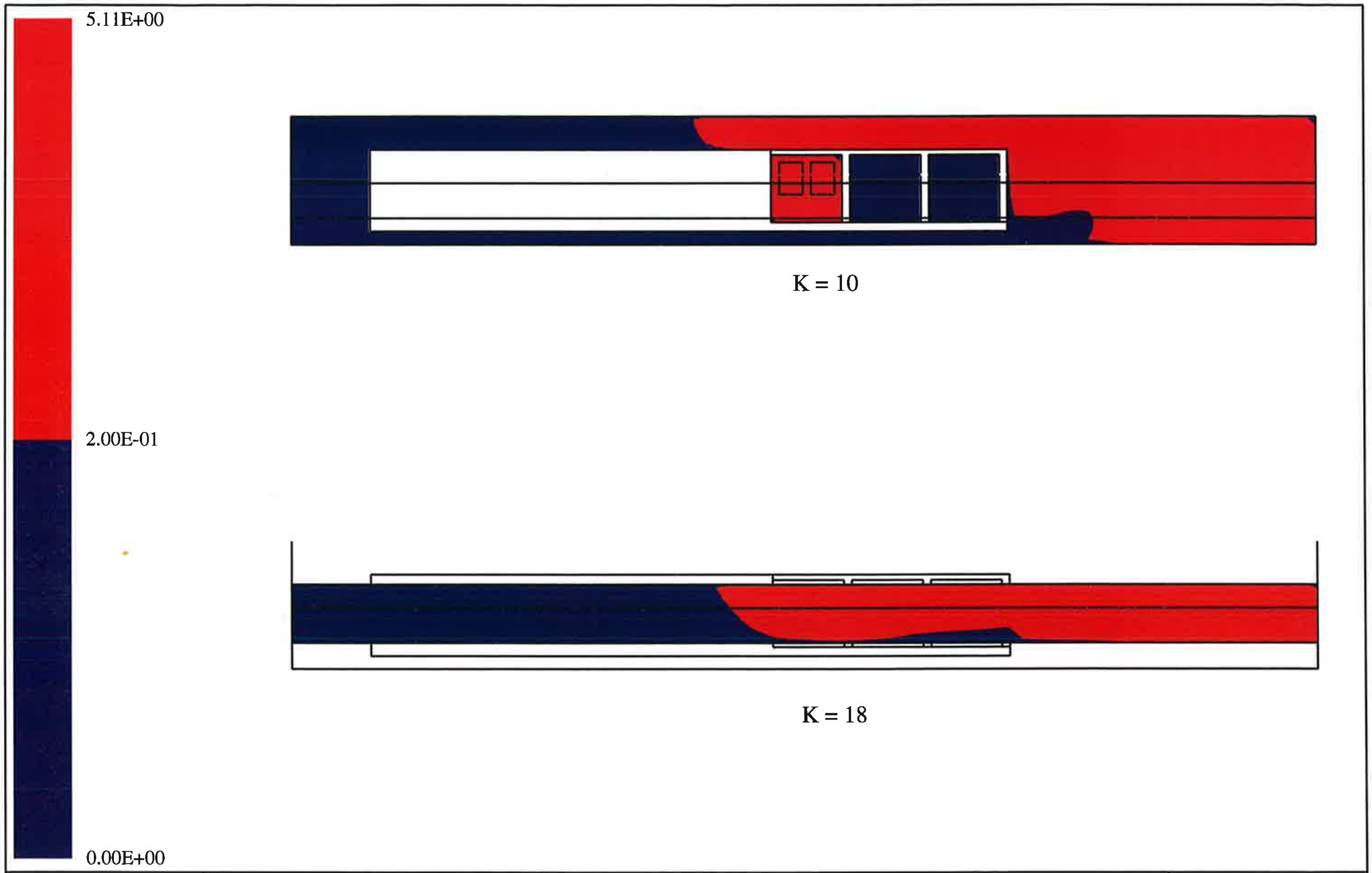
3-A-SS-SM2-PI
Attenuation (1/m)
Scenario 1

Figure 4b
Parsons
Brinckerhoff



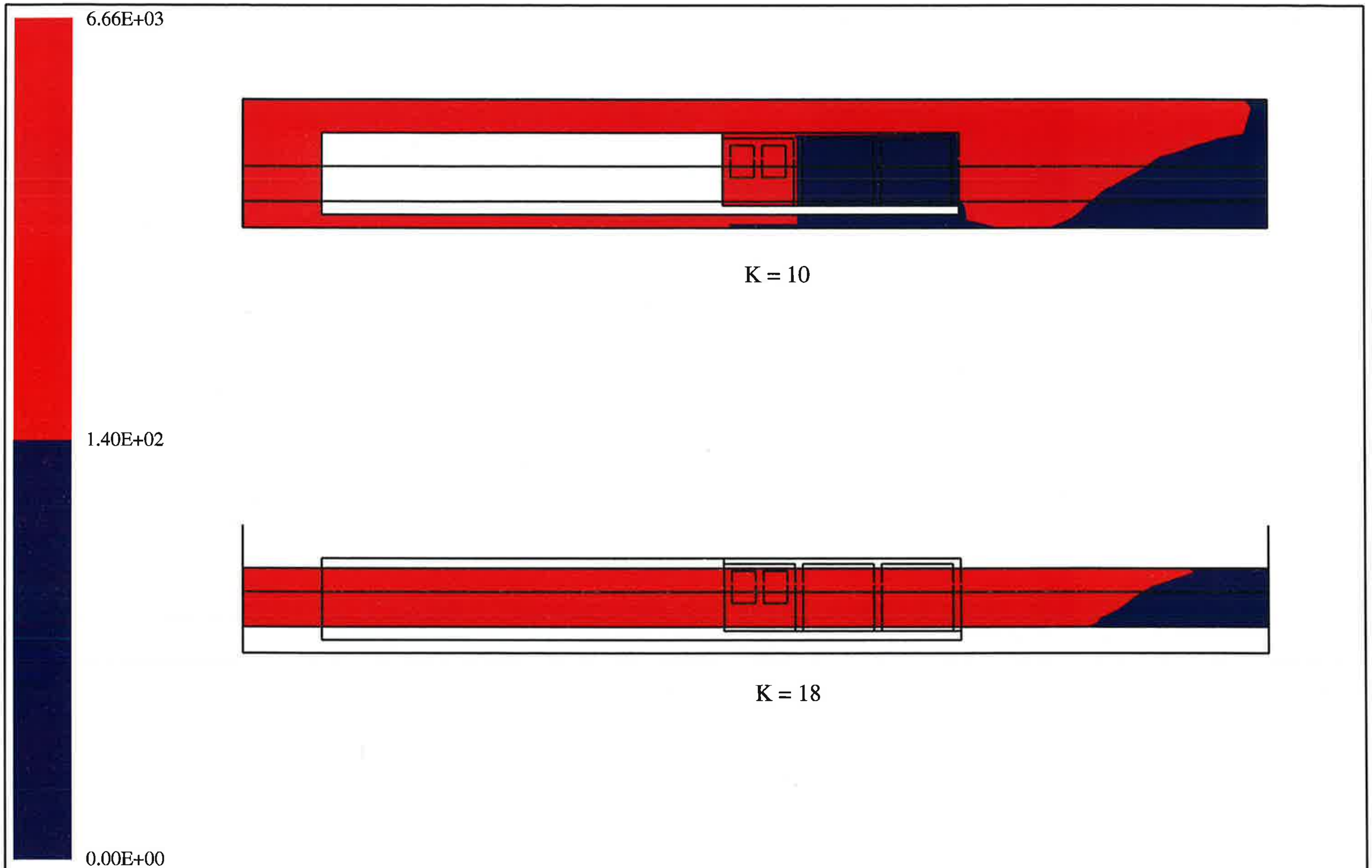
4-A-SS-SM3-FI
Temperature (Deg F)
Scenario 1

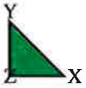
Figure 5 a
Parsons
Brinckerhoff



4-A-SS-SM3-FI
Attenuation (1/m)
Scenario 1

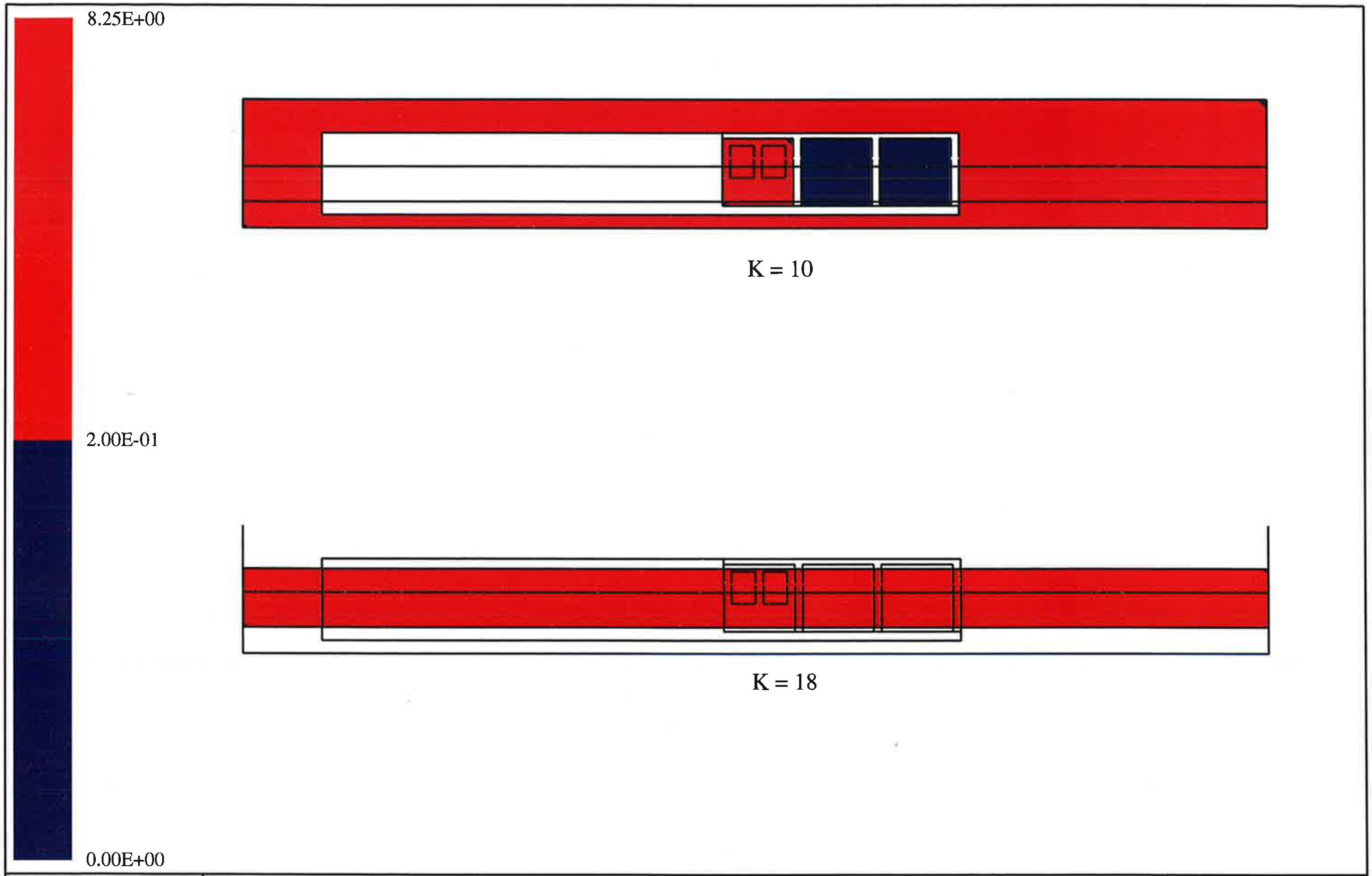
Figure 5b
Parsons
Brinckerhoff

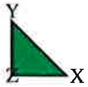




5-B-SS-SM3-FI
Temperature (Deg F)
Scenario 2

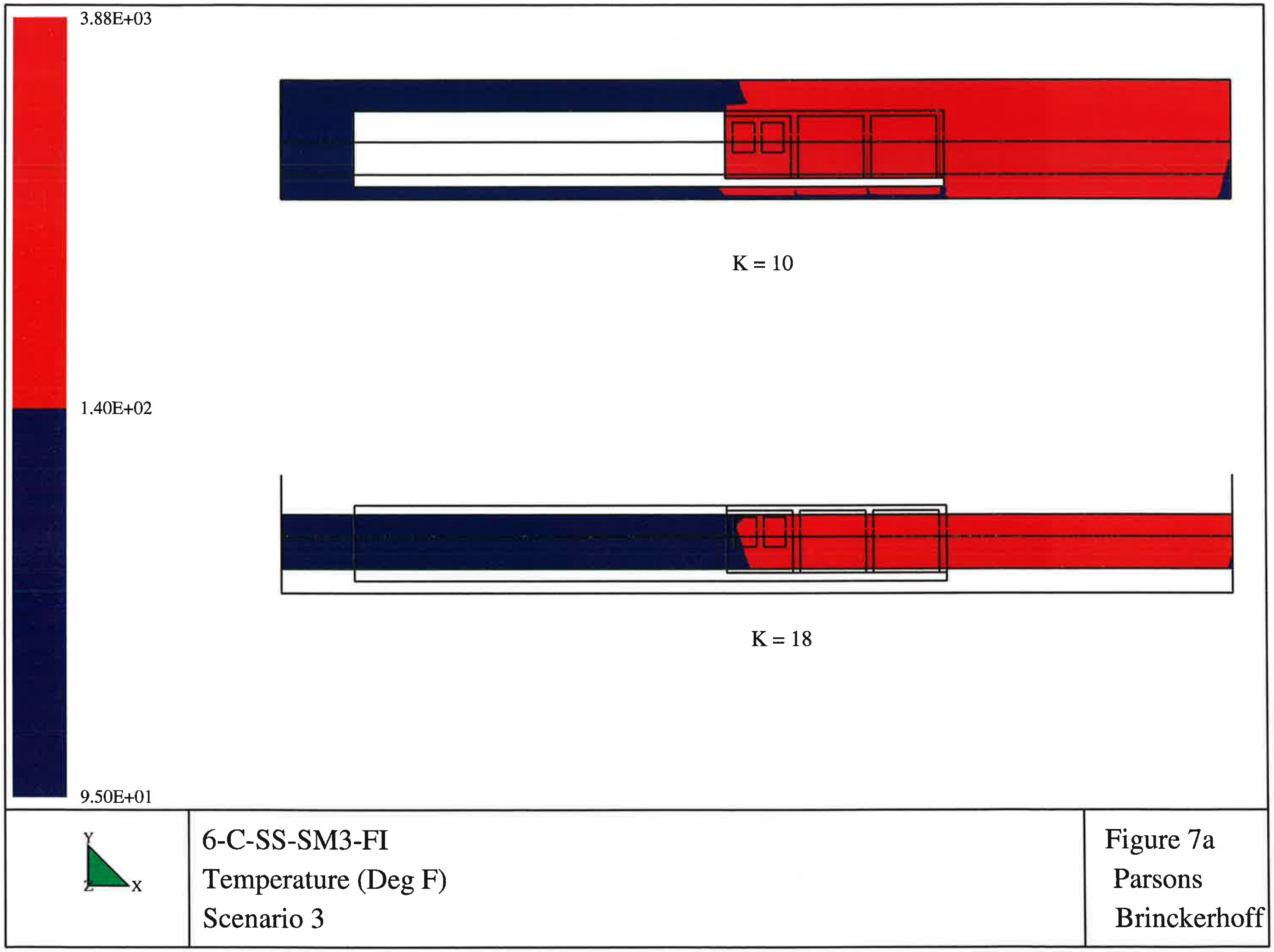
Figure 6a
Parsons
Brinckerhoff





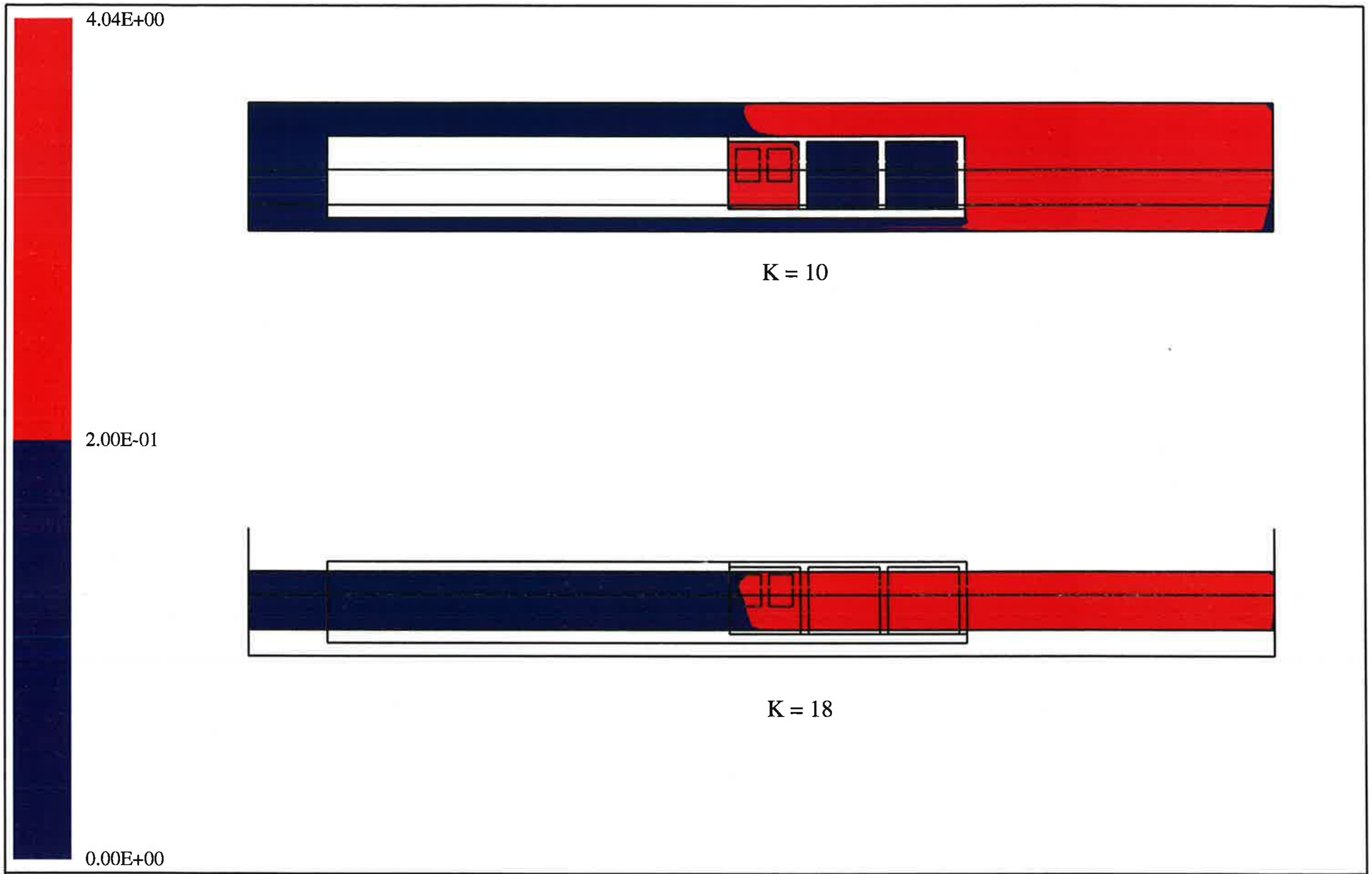
5-B-SS-SM3-FI
Attenuation (1/m)
Scenario 2

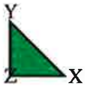
Figure 6b
Parsons
Brinckerhoff



6-C-SS-SM3-FI
Temperature (Deg F)
Scenario 3

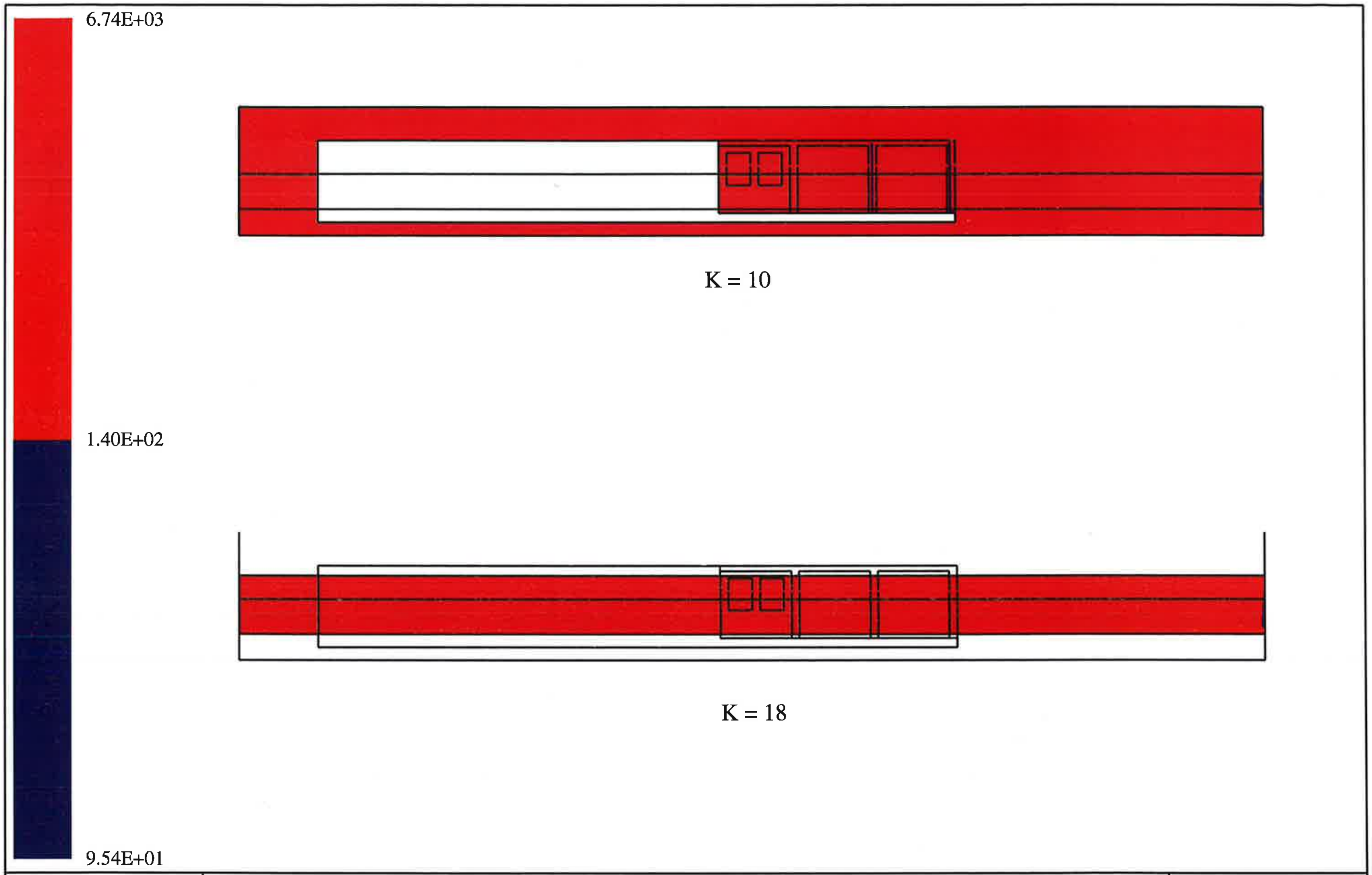
Figure 7a
Parsons
Brinckerhoff





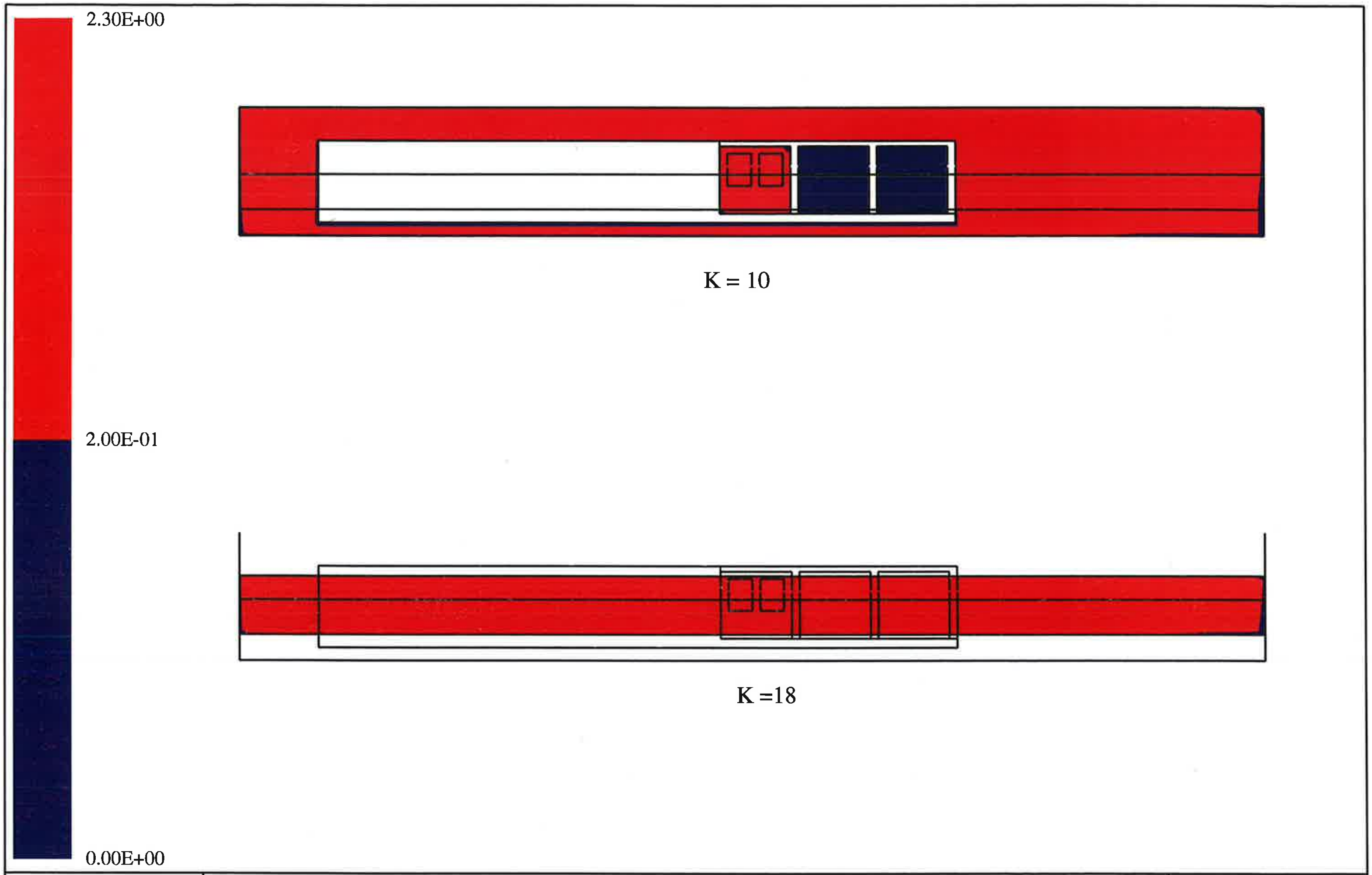
6-C-SS-SM3-FI
Attenuation (1/m)
Scenario 3

Figure 7b
Parsons
Brinckerhoff



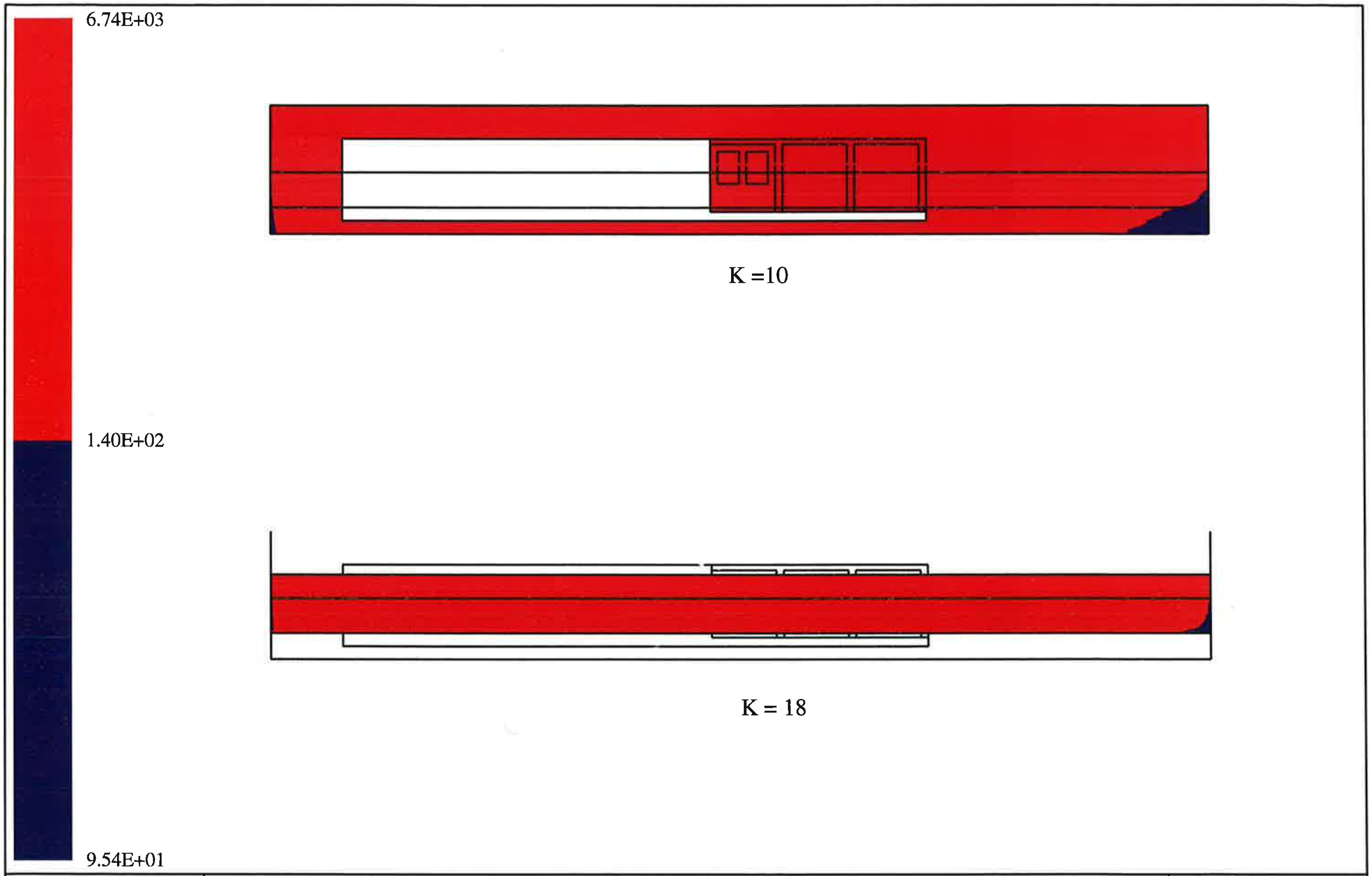
7-D-SS-SM3-FI (Flow Rate = 16232 cfm)
Temperature (Deg F)
Scenario 4

Figure 8a
Parsons
Brinckerhoff



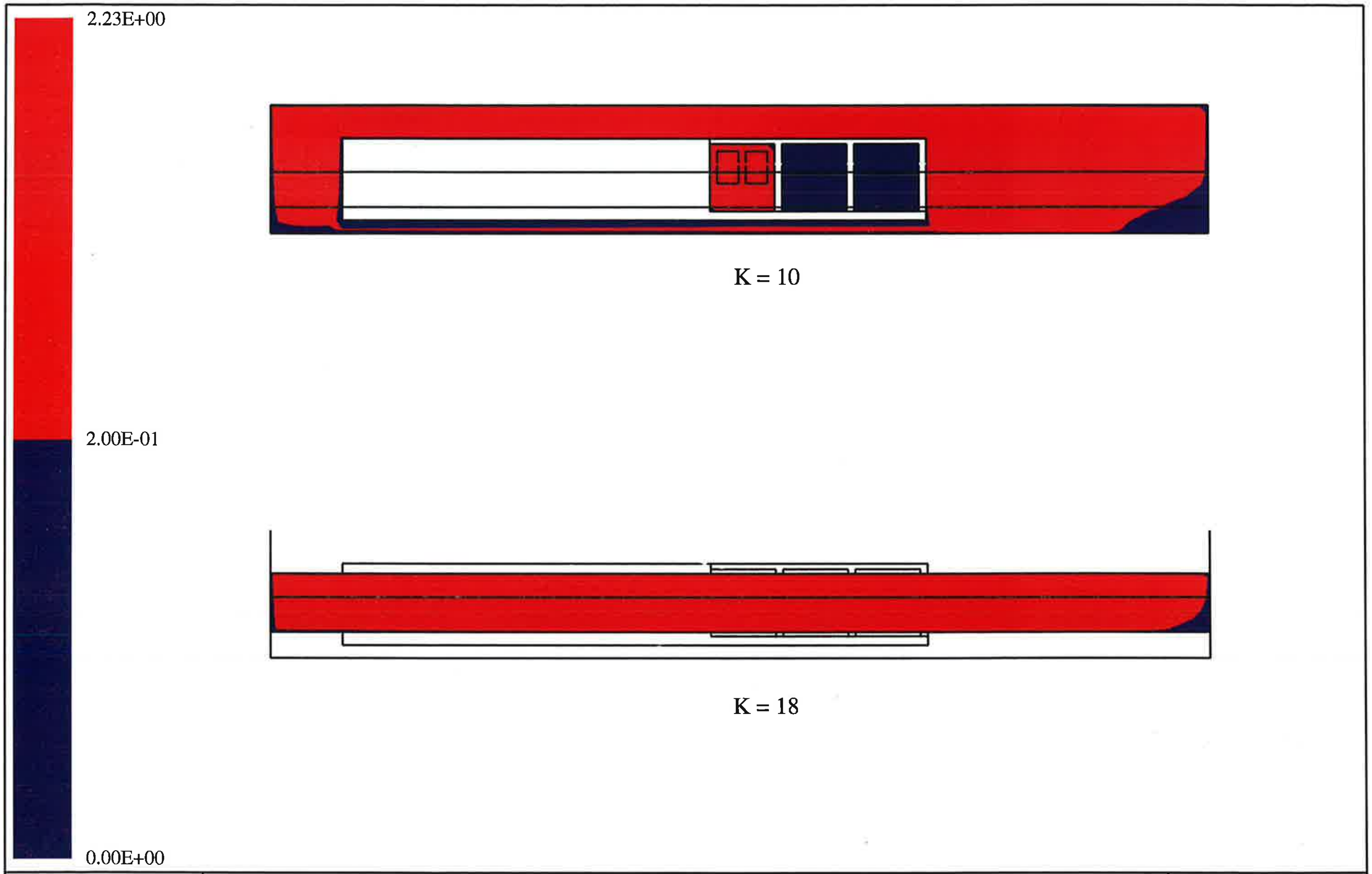
7-D-SS-SM3-FI (Flow Rate = 16232 cfm)
Attenuation (1/m)
Scenario 4

Figure 8b
Parsons
Brinckerhoff



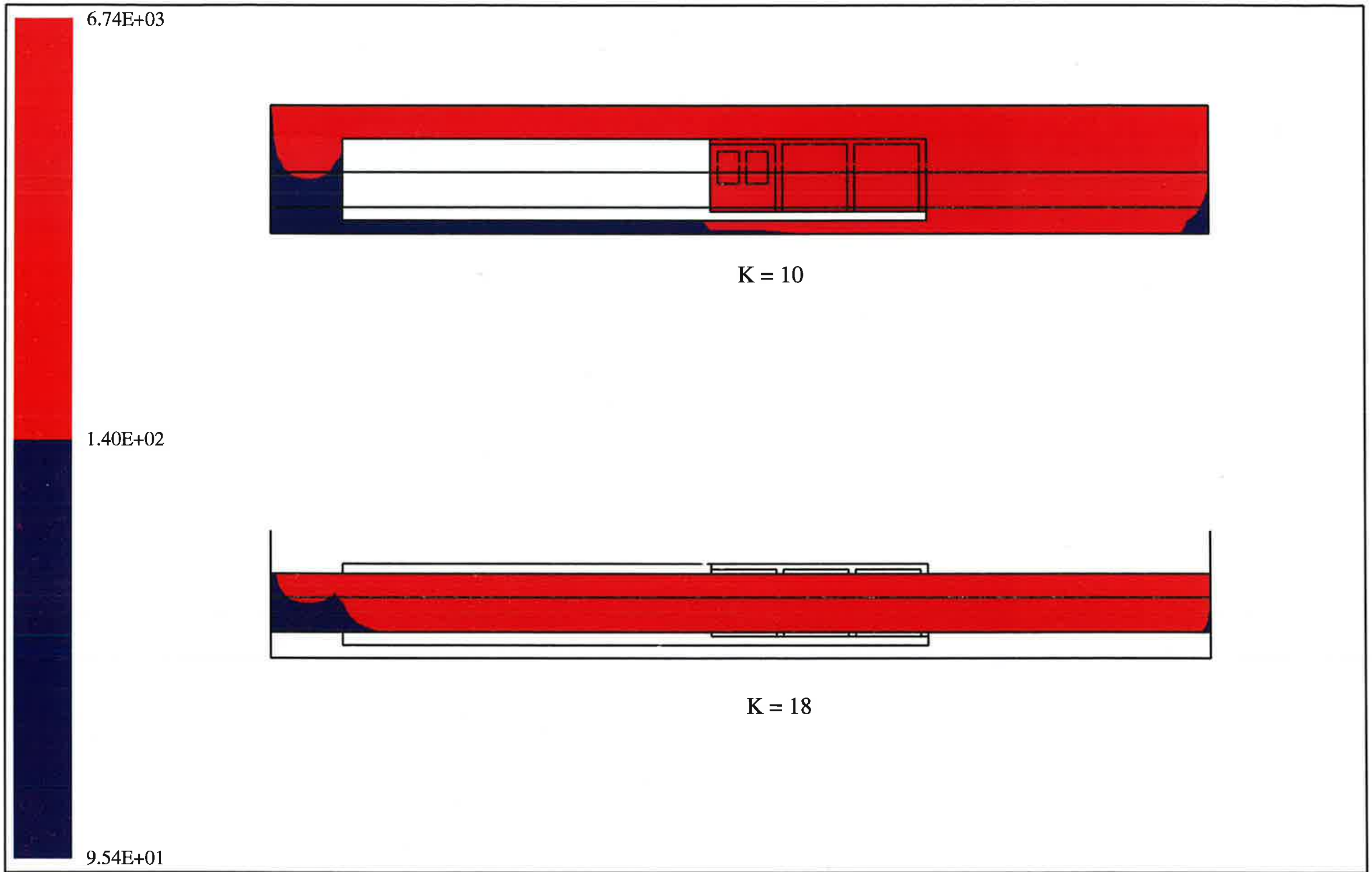
8-D-SS-SM3-FI (Flow Rate = 25000 cfm)
Temperature (Deg F)
Scenario 4

Figure 9a
Parsons
Brinckerhoff



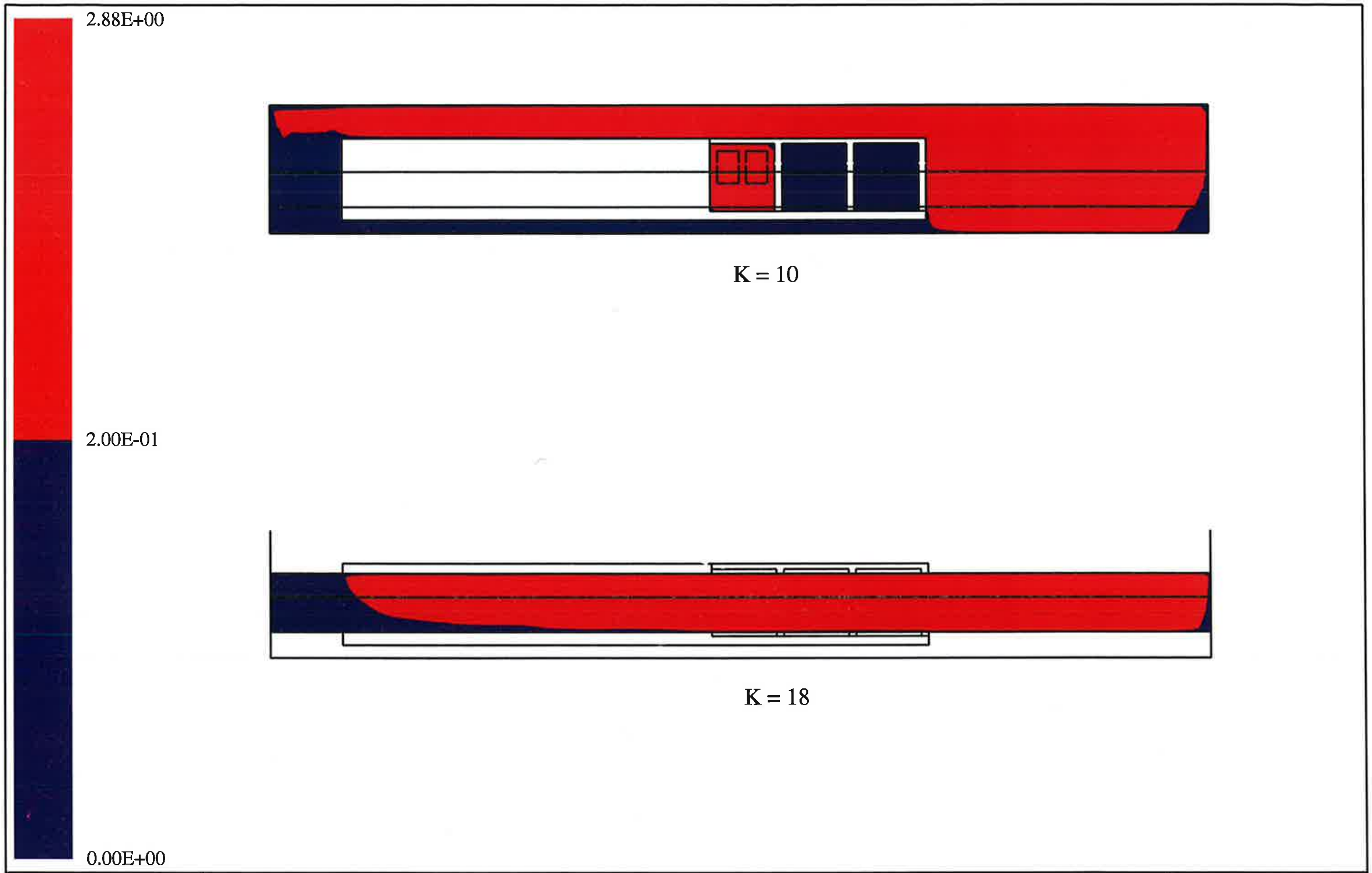
8-D-SS-SM3-FI (Flow Rate = 25000 cfm)
Attenuation (1/m)
Scenario 4

Figure 9b
Parsons
Brinckerhoff



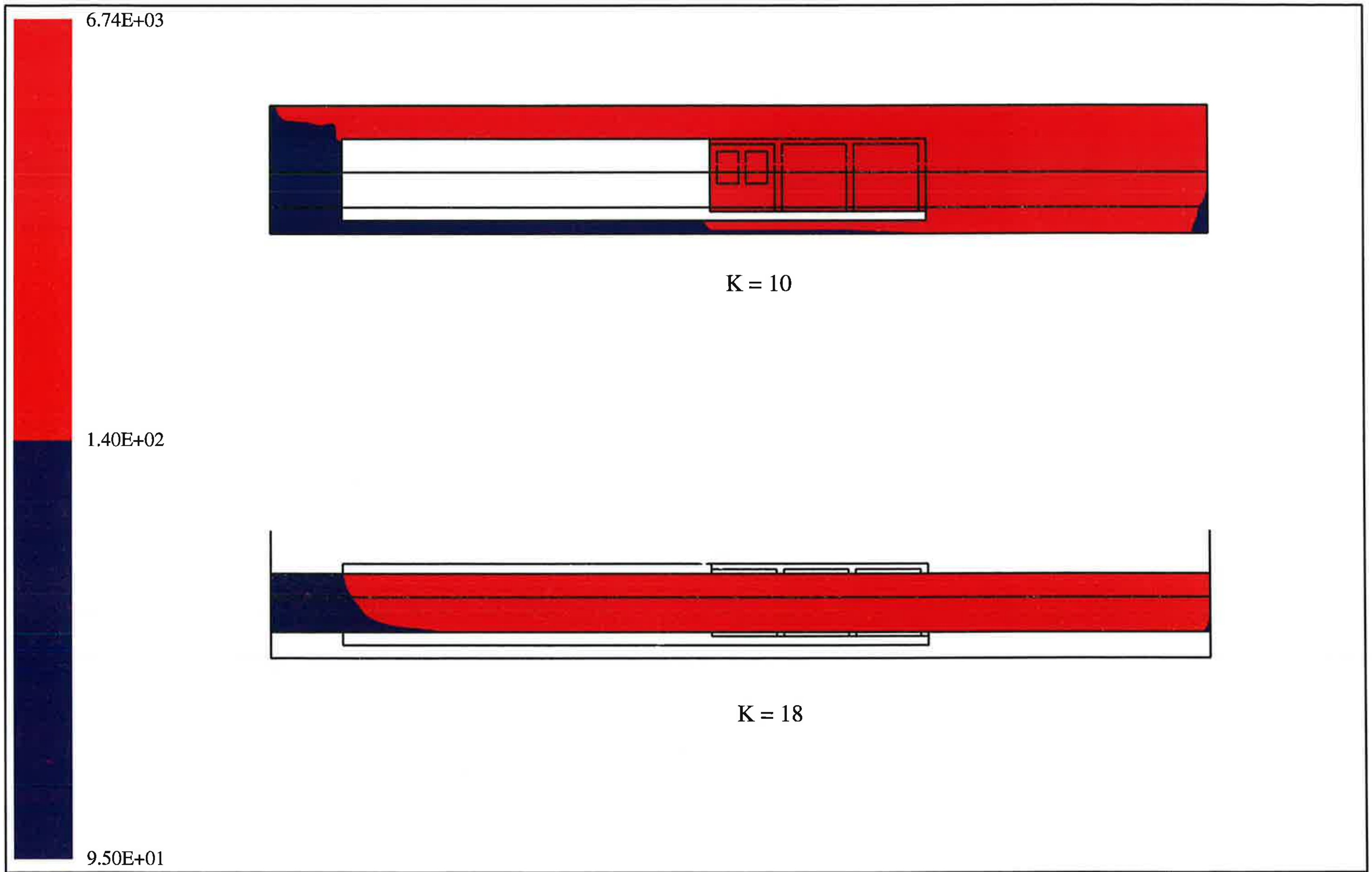
9-D-SS-SM3-FI (Flow Rate = 35000 cfm)
Temperature (Deg F)
Scenario 4

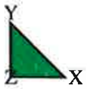
Figure 10a
Parsons
Brinckerhoff



9-D-SS-SM3-FI (Flow Rate = 35000 cfm)
Attenuation (1/m)
Scenario 4

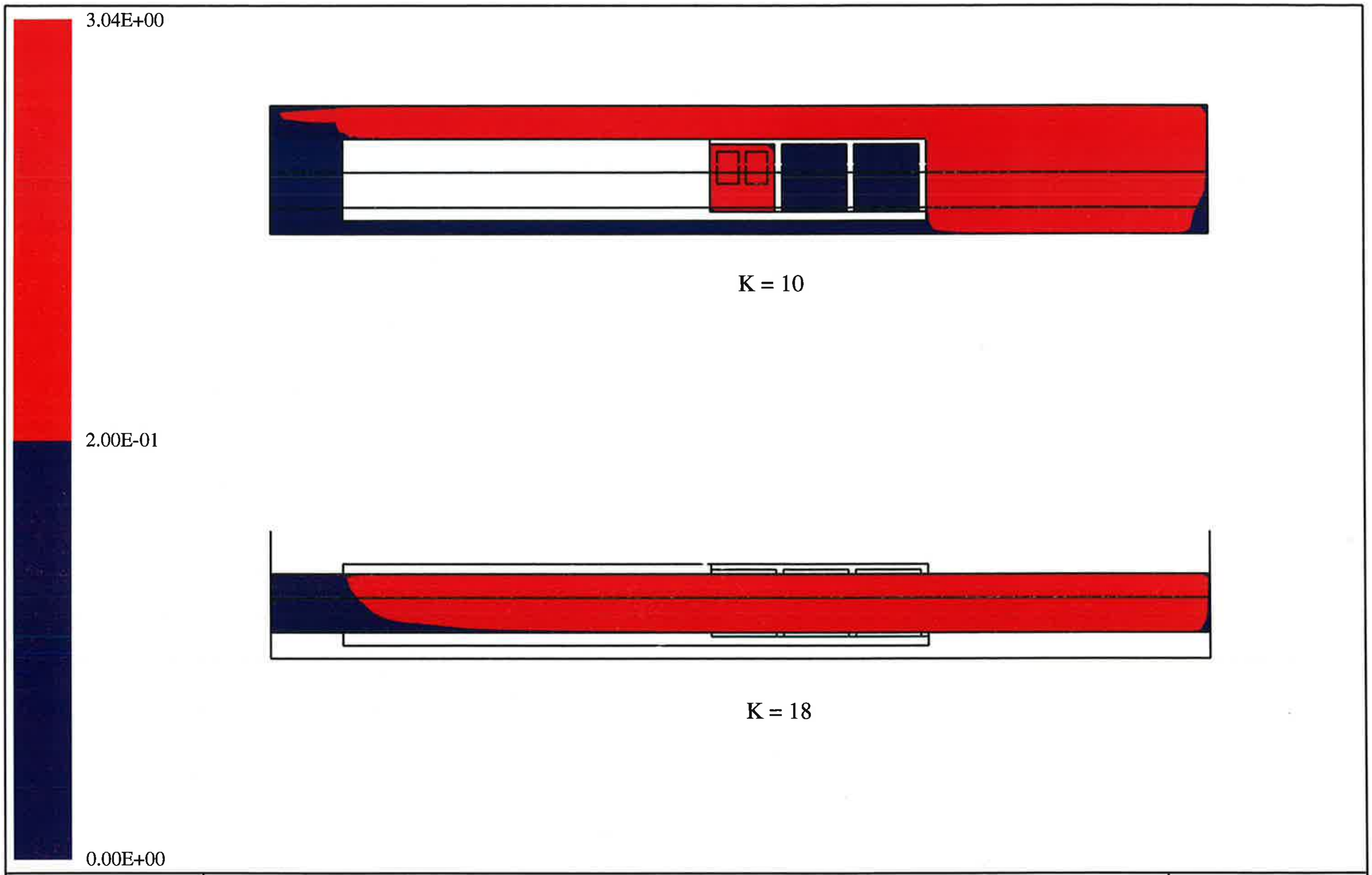
Figure 10b
Parsons
Brinckerhoff

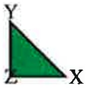




10-D-SS-SM3-FI (Flow Rate = 40000 cfm)
Temperature (Deg F)
Scenario 4

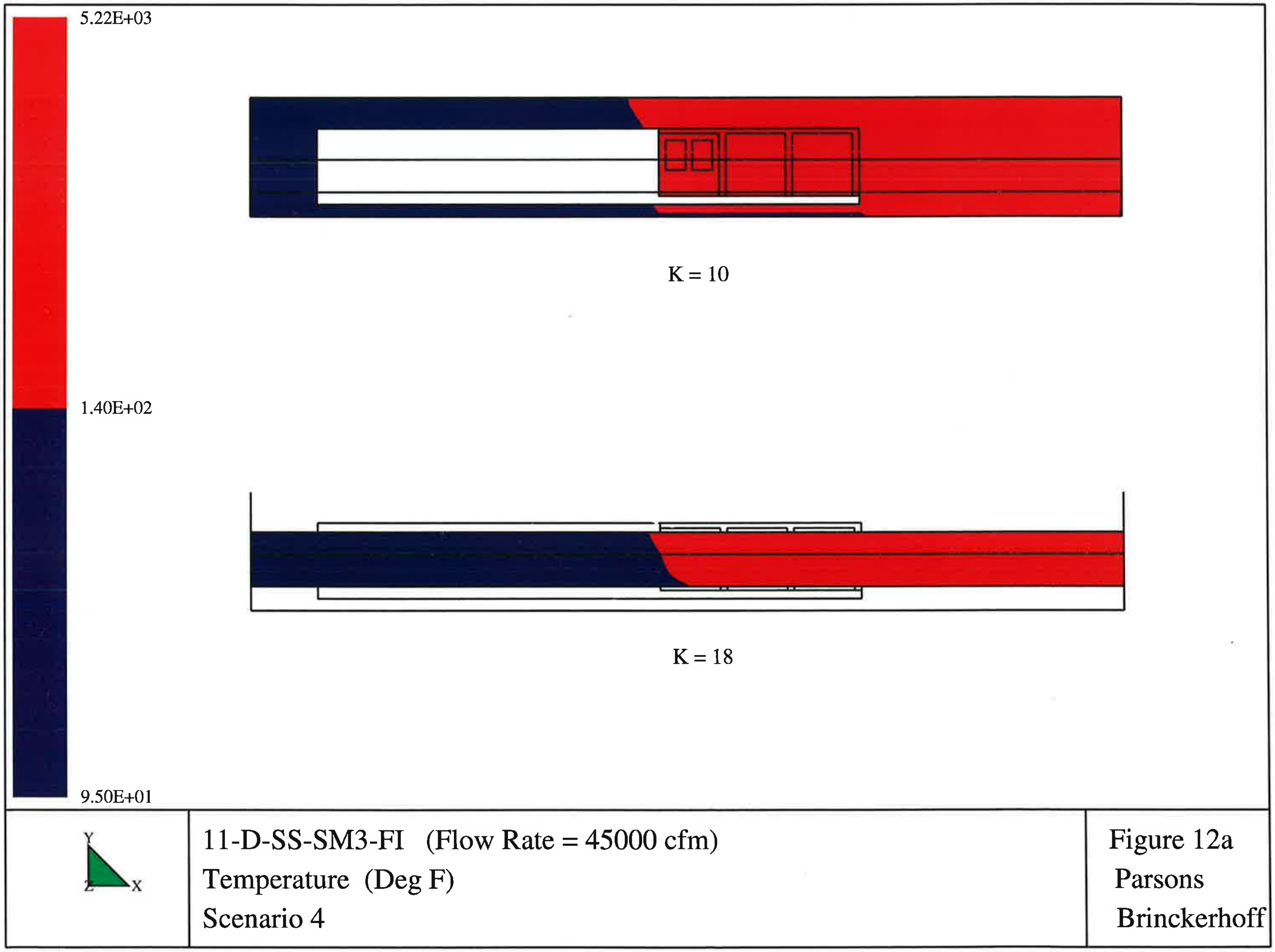
Figure 11a
Parsons
Brinckerhoff





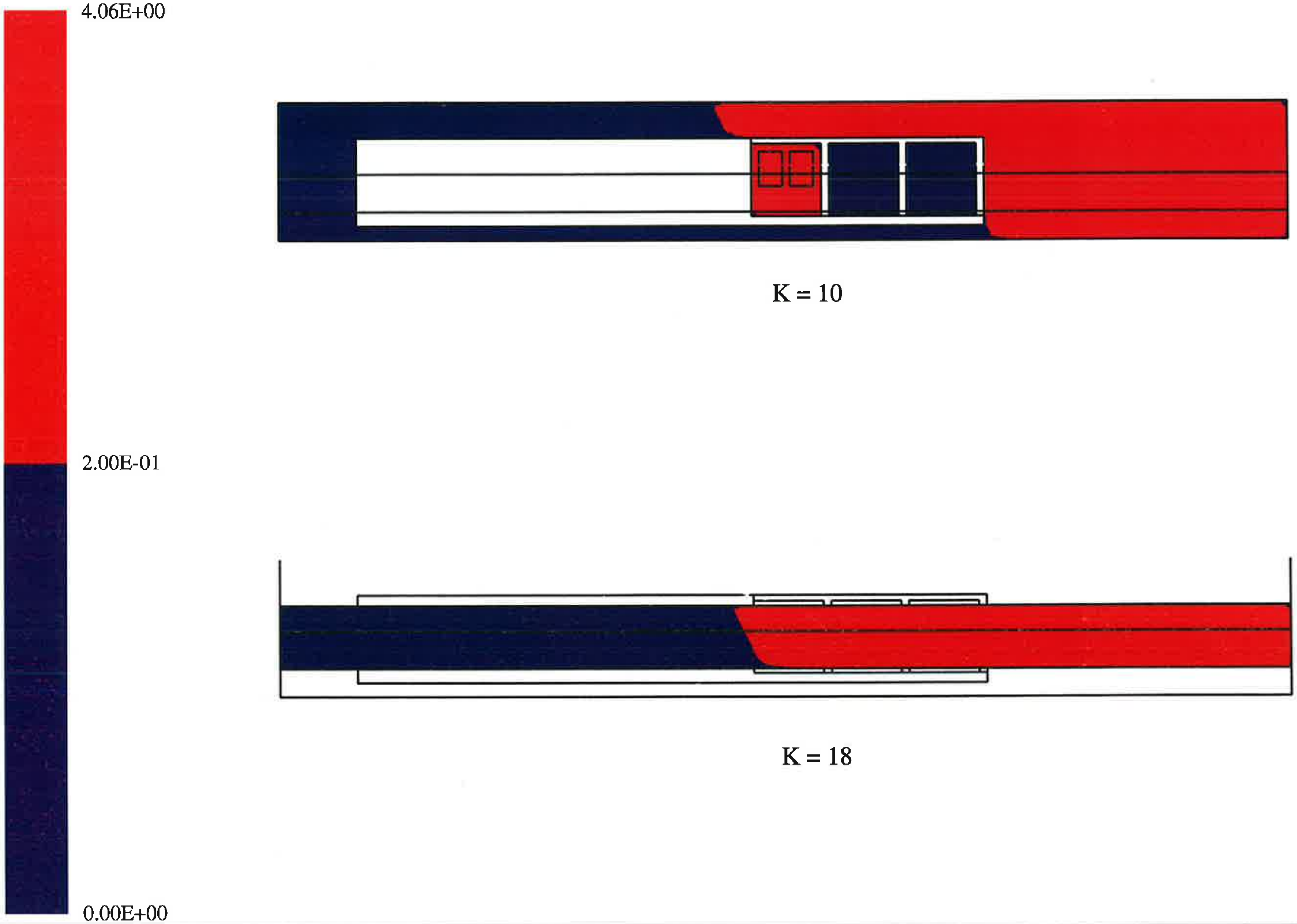
10-D-SS-SM3-FI (Flow Rate = 40000 cfm)
Attenuation (1/m)
Scenario 4

Figure 11b
Parsons
Brinckerhoff



11-D-SS-SM3-FI (Flow Rate = 45000 cfm)
Temperature (Deg F)
Scenario 4

Figure 12a
Parsons
Brinckerhoff



11-D-SS-SM3-FI (Flow Rate = 45000 cfm)
Attenuation (1/m)
Scenario 4

Figure 12b
Parsons
Brinckerhoff