CFAST – Consolidated Model of Fire Growth and Smoke Transport (Version 5)

User's Guide

Richard D. Peacock
Walter W. Jones
Glenn P. Forney

CFAST – Consolidated Model of Fire Growth and Smoke Transport (Version 5)

User's Guide

Richard D. Peacock
Walter W. Jones
Glenn P. Forney

Fire Research Division
Building and Fire Research Laboratory

December 2004

U.S. Department of Commerce
Donald L. Evans, Secretary

Technology Administration
Phillip J. Bond, Under Secretary for Technology

National Institute of Standards and Technology
Arden L. Bement, Jr., Director

Bibliographic Information

Abstract

CFAST is a two-zone fire model capable of predicting the environment in a multi-compartment structure subjected to a fire. It calculates the time evolving distribution of smoke and gaseous combustion products as well as the temperature throughout a building during a user-prescribed fire. This report describes the use of the model, including installing and running the software, the computer platforms upon which it is supported and examples to verify correct installation.

DISCLAIMER

The U. S. Department of Commerce makes no warranty, expressed or implied, to users of - CFAST and associated computer programs, and accepts no responsibility for its use. Users of CFAST assume sole responsibility under Federal law for determining the appropriateness of its use in any particular application; for any conclusions drawn from the results of its use; and for any actions taken or not taken as a result of analyses performed using these tools.

Users are warned that CFAST is intended for use only by those competent in the field of fire safety and is intended only to supplement the informed judgment of the qualified user. The software package is a computer model which may or may not have predictive value when applied to a specific set of factual circumstances. Lack of accurate predictions by the model could lead to erroneous conclusions with regard to fire safety. All results should be evaluated by an informed user.

INTENT AND USE

The algorithms, procedures, and computer programs described in this report constitute a methodology for predicting some of the consequences resulting from a prescribed fire. They have been compiled from the best knowledge and understanding currently available, but have important limitations that must be understood and considered by the user. The program is intended for use by persons competent in the field of fire safety and with some familiarity with personal computers. It is intended as an aid in the fire safety decision-making process.

Contents

Tables

CFAST – Consolidated Model of Fire Growth and Smoke Transport (Version 5) User's Guide

Richard D. Peacock, Walter W. Jones, and Glenn P. Forney

Fire Research Division
Building and Fire Research Laboratory

1 Introduction

CFAST is a two-zone fire model used to calculate the evolving distribution of smoke, fire gases and temperature throughout compartments of a building during a fire. These can range from very small containment vessels, on the order of 1 m^3 to large spaces on the order of 1000 m^3. This guide describes how to obtain the model, verify its correct installation, create input data in an appropriate form, and analyze of the output of a simulation.

The modeling equations used in CFAST take the mathematical form of an initial value problem for a system of ordinary differential equations (ODEs). These equations are derived using the conservation of mass, the conservation of energy (equivalently the first law of thermodynamics), the ideal gas law and relations for density and internal energy. These equations predict as functions of time quantities such as pressure, layer height and temperatures given the accumulation of mass and enthalpy in the two layers. The CFAST model then consists of a set of ODEs to compute the environment in each compartment and a collection of algorithms to compute the mass and enthalpy source terms required by the ODEs. The formulation of the equations, their solution, and discussion of validation and verification of the code are presented in a companion document [1].

All of the data to run the model is contained in a primary data file, together with databases for objects, thermophysical properties of boundaries, and sample prescribed fire descriptions [2]. These files contain information about the building geometry (compartment sizes, materials of construction, and material properties), connections between compartments (horizontal flow openings such as doors, windows, vertical flow openings in floors and ceilings, and mechanical ventilation connections), fire properties (fire size and species production rates as a function of time), and specifications for detectors, sprinklers, and targets (position, size, heat transfer characteristics, and flow characteristics for sprinklers). Materials are defined by their thermal conductivity, specific heat, density, thickness, and burning behavior. Throughout the discussion on the model inputs, notes are included to provide additional insight on the model's operation.

The output of CFAST are the sensible variables that are needed for assessing the environment in a building subjected to a fire. These include temperatures of the upper and lower gas layers within each compartment, the ceiling/wall/floor temperatures within each compartment, the visible smoke and gas species concentrations within each layer, target temperatures and sprinkler activation time.

Many of the outputs from the CFAST model are relatively insensitive to uncertainty in the inputs for a broad range of scenarios. However, the more precisely the scenario is defined, the more accurate the results will be. Not surprisingly, the heat release rate is the most important variable because it provides the driving force for fire-driven flows. Other variables related to compartment geometry such as compartment height or vent sizes, while important for the model results, are typically more easily defined for specific design scenarios than fire related inputs.

The first public release of CFAST was version 1.0 in June of 1990. This version was restructured from FAST [3] to incorporate the "lessons learned" from the zone model CCFM [4], namely that modifications and additions to the model are easier and more robust if the components such as the physical routines are separated from the solver code used by the model. Version 2 was released as a component of Hazard 1.2 in 1994 [5]. The first of the 3.x series was released in 1995 and included a vertical flame spread algorithm, ceiling jets and nonuniform heat loss to the ceiling, spot targets, and heating and burning of multiple objects in addition to multiple prescribed fires. Ignition was assigned based on a critical heat flux, a critical temperature, or a critical time input by the user. As CFAST evolved over the next five years, version 3 included smoke and heat detectors, suppression through heat release reduction, better characterization of flow through doors and windows, vertical heat conduction through ceiling/floor boundaries, and non-rectangular compartments. In 2000, version 4 was released and included horizontal heat conduction through walls, and horizontal smoke flow in corridors. Version 5 improved the combustion chemistry. The current version is 5.1.

The rest of this guide is organized as follows:

- Chapter 2 describes how to obtain, set up, and install CFAST. Hardware requirements are included.
- Chapter 3 summarizes how to create an input file for a desired simulation, run the program on a Windows PC and monitor progress of the simulation while the program is running. Software implementation limits and error messages from the program are described.
- Chapter 4 details all of the inputs to the model. Default values and limits for the inputs are included. Notes and limitations on the use of the model are presented where appropriate.
- Chapter 5 includes the model outputs and includes a description of each output and its engineering units.

2 Getting Started

The software for the CFAST model is a Fortran 90 computer program that predicts the environment within compartmented structures resulting from a fire prescribed by the user. Details of the calculations are described in reference [1].

2.1 How to Get CFAST

All of the files associated with CFAST can be obtained at:

```
http://fire.nist.gov/cfast
```

Information about new versions, bug fixes, and documentation for the model and software are available on this web site. The CFAST distribution consists of a self-extracting set-up program for Windows-based PCs. After downloading the set-up program to a PC, double-clicking on the file's icon walks the user through a series of steps for installation of the program. The most important part of the installation is the creation of a directory (called c:\nist\cfast5 by default) in which the CFAST executable files, sample cases, and supplemental data files are installed.

CFAST is documented by two publications, this user's guide and a technical reference guide [1]. The technical reference guide describes the underlying physical principles, provides a comparison with other models and includes a description of the limitations of this zone model. This user's guide describes how to use the model and applies to version 5 and later. All the documentation is available on the web site.

2.2 Computer Hardware Requirements

CFAST requires a relatively fast processor and a sufficient amount of random-access memory (RAM) for complex cases. The processor should be at least as fast as a 1 GHz Pentium III, with at least 256 MB of memory for complex cases. Typical calculation times for a 2 compartment scenario can range from a few seconds to multiple hours, depending on the details of the scenario. Plus, a large hard drive is needed to store the output of calculations. It is not uncommon for a single calculation to generate output files as large as 1 GB.

2.3 Computer Operating System and Software Requirements

CFAST runs in a command prompt in Microsoft Windows. The latest version of the model has been tested on both Windows 2000 and Windows XP, but should also run on any version beginning with Windows 95.

2.4 Verifying Correct Installation and Operation

Sample input files are provided with the program for new users who are encouraged to first run the sample calculations before attempting to create an input file. To run the model, open a command prompt window (`<START>/<RUN> cmd`, in Windows XP) and change directories ("`cd`") to the installation directory, for example, `\NIST\CFAST511`. Then run the program with the command

```
cfast
```

This runs a very simple test case and it should be completed quickly. Additional details on running CFAST are included in the next chapter. To verify that the installation has been done correctly, the first few lines at the beginning and end of the output of the model should appear as

```
**      CFAST  Version   5.1.1  Run 10/1/2004     **
**                                                **
**           A contribution of the        **
** National Institute of Standards and Technology **
**           Gaithersburg, MD  20899        **
**             Not subject to Copyright       **

...

Time =   1800.0 seconds.

Compartment   Upper   Lower   Inter.  Pyrol    Fire       Pressure  Ambient
              Temp.   Temp.   Height  Rate     Size                 Target
              (K)     (K)     (m)     (kg/s)   (W)        (Pa)      (W/m^2)
-------------------------------------------------------------------------
    1         377.0   303.8   0.72    7.249E-03 1.050E+05 -9.484E-02  437.
Outside                                         0.00
```

The date will change according to when the calculation was run. This case checks several attributes of the installation including the presence of the database files (see auxiliary files in the section on building input files). Additional explanation of the results of this run are described in Chapter 5.

To further verify correct installation, the following commands can be used:

```
CFAST standard.dat standard.lst /r:winfs
REPORT standard.hi standard.rs
REPORTSS standard.hi standard.csv /r:winfs
```

This will generate the files `standard.hi, standard.lst, standard.rs` and `standard.csv`. There is a set of equivalent files included with the distribution (named `standardd.lst, standardd.rs,` and `standardd.csv` and are in the data directory of the standard installation, typically located at `c:\nist\cfast511\data`), which can be compared to the installation test results.

3 Running CFAST

Running CFAST is relatively simple. All of the parameters that describe a given fire scenario are typed into a text file that is referred to as the "data" or "input" file. In this document, the data file is designated as `filename.dat`, where "filename" stands for any character string that helps to identify the simulation. All of the output files associated with the calculation would typically have this common prefix. In addition to the input file, there are often several external files containing input parameters for the simulation. These files are referred to as "database" files and contain parameters describing common materials and fuels.

It is suggested that a new user start with an existing data file, run it as is, and then make the appropriate changes to the input file for the desired scenario. By running a sample case, the user becomes familiar with the procedure and ensures that his/her computer is up to the task before embarking on learning how to create new input files.

3.1 Creating a CFAST Input Data File

All of the data to run the model is contained in an input data file. Also needed are databases for objects, thermophysical properties of boundaries, and sample prescribed fire descriptions provided with the model [6]. These files contain information about the building geometry (compartment sizes, materials of construction, and material properties), connections between compartments (horizontal flow openings such as doors, windows), vertical flow openings in floors and ceilings, and mechanical ventilation connections), fire properties (fire size and species production rates as a function of time), and specifications for detectors, sprinklers, and targets (position, size, heat transfer characteristics, and flow characteristics for sprinklers). Materials are defined by their thermal conductivity, specific heat, density, thickness, and burning behavior (heat release rate, ignition properties, and species yields).

The input data file provides the program with parameters to describe the scenario under consideration. The parameters are organized into groups of related variables. Each line of the input data file contains inputs related to a single group and begins with a keyword that identifies the input. For example, compartment geometry is described by a set of lines that define the width (keyword `WIDTH`), depth (keyword `DEPTH`), and height (keyword `HEIGH`) of each compartment. A description of the input parameters can be found in Chapter 4.

3.2 File Naming and Location

By default, the CFAST installation places all program files in the directory `c:\NIST\CFAST511` and sample input data files in the directory `c:\NIST\CFAST511\DATA`. While these locations can be changed during installation, the documentation in this user's guide assumes these locations.

File naming conventions in CFAST limit the length of file names to the DOS standard of 8 characters plus a three character extension such as `standard.dat`. By default, all input data files are assumed located in the data directory at `c:\NIST\CFAST511\DATA` and all output files are written to this directory as well. The location of these files can only be changed by editing the configuration file, below.

In addition, there are several files that CFAST uses to communicate with its environment. They include 1) an input data file, required for every simulation, 2) a configuration file, 3) the thermal properties database, 4) the database of additional fire objects, 5) a history file, and 6) a restart file. Documentation of the input data file is included as chapter 4 of this user's guide. The format of the configuration file, thermal database and objects database are detailed in Appendix A and Appendix B. Specification of the history and restart file are included in section 4.2.3.

There are defaults for the configuration file, thermal physical database and database of additional fire objects. The default configuration file is `HV5.CF`. In this file there is a default data file. If CFAST is run without arguments, the program will look for `HV5.CF` and run the data file specified in the configuration file. The standard distribution specifies `standard.dat` as the default data file. The initial instructions for installing and testing the model for correct installation implies running this test case. Since it is only a 10 second simulation, the scenario runs very quickly.

3.3 Starting a CFAST Calculation

CFAST is run from a Windows command prompt. To run the model, open a command prompt window (`<START>/<RUN> cmd`) and change directories ("`cd`") to the installation directory, for example, \NIST\CFAST511. Then run the program with the command:


```
CFAST <input file> <output file> <options>
```

where

<input file> is the name of the input data file that defines the simulation.

<output file> is the name of the file to which any specified text output will be written. If omitted, the text output will go directly to the computer screen.

<options> specify any output reporting or other options to control the output of the program. The options available are as follows:

/k – directs that CFAST should ignore any input from the keyboard. This option is useful mainly for running a number of test cases in batch mode

/l – directs that CFAST should generate an error log file named CFAST.LOG which contains detailed debugging information.

/n – directs that CFAST should not print out the normal header to the console at the beginning of a simulation. This option is useful mainly for running a number of test cases in batch mode

/r – directs cfast to includes additional variables in any text or spreadsheet output. The /r is followed by a colon and one or more of the following characters to indicate the level of output:

> w – include wall surface and target fluxes in the output
> i – include model initial conditions and scenario description in the output
> n – include "normal" printout of layer temperatures, interface height, and fire size in the output
> f – include vent flows in the output
> s – include layer and wall surface species concentrations in the output

for example, the command:

```
cfast filename.dat filename.lst /r:winfs
```

would run the program using the input file `filename.dat`, with text output going to the file `filename.lst`, and include the most complete text or spreadsheet output.

3.4 Monitoring Calculation Progress

Two keys allow some interaction with the model while it runs. The first is the `Esc` key. When pressed, the key terminates the run. The second key is the `<f5>` function key. Pressing the `<f5>` function key causes the current simulation time and last time step to print to the text output. When the text output goes to the screen the user can see the progress of the model. When the model runs with an output file as in the example, the information prints in the file.

For example, pressing `<f5>` during the run might produce:

```
Time =     9.58550207410571      DT =    0.353489940990121
```

The output means the model has completed about 9.59 s of simulated time and the last completed time step was about 0.353 s long. The first second of simulation time takes a while to run because of the start up of the solver, so the <f5> function key doesn't provide a lot of information. After the first second, time steps larger than 0.2 s mean the model is having little problem solving the differential equations and is moving very quickly. Times larger than 0.1 s indicate

7

the model is moving through the simulation at a sufficient speed. Steps sizes less than 0.01 seconds indicate that the solution of the differential equations is more difficult. While the simulation will take longer to complete with such small time steps, precision of the resulting calculations is unaffected.

3.5 Model Output Options

In addition to direct text output from the model, the programs `TOSMOKEVIEW`, `REPORT` and `REPORTSS` can be used to generate output after the simulation is complete. `TOSMOKEVIEW` is used to send data to the `smokeview` program for viewing output in three dimensional form [7]. `REPORT` and `REPORTSS` generate text output, either as a normal text output in `REPORT`, or in spreadsheet form in `REPORTSS`. All three programs use a binary output file named by the user and created by CFAST during the simulation.

3.5.1 Graphical Output: How to Run TOSMOKEVIEW:

The program, `TOSMOKEVIEW`, uses only the history file generated by the model and the name is given in the DUMPR keyword. For example, if a snippet from a data files called STANDARD.DAT looked like:

```
DUMPR STANDARD.HI
```

then the "plt" and "smv" files for `smokeview` would be generated by:

```
TOSMOKEVIEW STANDARD
```

Please note that the extension is not used, and "hi" is assumed. There are no options for this program. It generates the "smv" and "plt" files for view in smokeview. For further information on using smokeview, please refer to its user's guide[7].

3.5.2 Text Output: How to Run REPORT and REPORTSS:

These two programs are in the form:

```
REPORT <input file> <output file> /r:<options> and
REPORTSS <input file> <output file> /r:<options>
```

Output options for REPORT and REPORTSS are the same as those for CFAST. Details of the outputs are included in Chapter 5.

REPORTSS outputs the same values as report. However, the format is a comma delimited file so that it can be easily imported into a spreadsheet. The caution is that REPORTSS writes all

available data to the spreadsheet which can result in more columns than spread sheets can handle. So it is recommended that only data of interest be written, rather than all options. The full set is /r:winfs, as with the normal printout. The recommendation would be to use /r:n, then /r:f and so on. This keeps the spread sheet to a manageable size.

The headings in the comma delimited file generally spell out the information that is in the column. Output units are consistent with those described in Chapter 5.

4 Setting Up the Input File for CFAST

4.1 Overview

CFAST is a computer program which uses an input file and generates one or more output files. The first step in performing a calculation is to generate a text input file that provides the program with all of the necessary information to describe the scenario under consideration. The most important inputs determine the geometry of the compartments in the scenario and the connections between these compartments. Next, the fire, detection, and suppression characteristics of the scenario are defined. Finally, there are a number of parameters that customize the output from the model. Each line of the file contains a keyword label that identifies the input, followed by a number of numerical or text inputs corresponding to the particular input keyword. A simple input file is shown below. This example is used in the discussion of the output in Chapter 5.

```
VERSN   5 Cable Tray Fire - Base Case
!Simulation time - 1800 seconds with printout every 120 and history file every 30 seconds
TIMES 1800  120       30
!These are the usual global configurations
TAMB  300    101300   0
EAMB  300    101300   0
!Put the history file to standard.hi - this is the usual naming convention
DUMPR STANDARD.HI
!Turn on the ceiling jet calculation
CJET  ALL
!A single compartment 9x5x4.5
HI/F   0
WIDTH 9.1
DEPTH 5
HEIGH 4.6
!Set thermophysical properties
CEILI GLASSFB3
WALLS CONCRETE
FLOOR CONCRETE
!Add a door which is open always open
HVENT  1 2 1 2.4   1.00 0 0 0 0
CVENT  1 2 1 1.00   1.00  1.00  1.00  1.00  1.00  1.00  1.00  1.00  1.00  1.00
CHEMI  16    50    12    24100000  300    493.15 0.3
!Put a fire in the first compartment
LFBO   1
LFBT   2
FPOS   3   5.7   0
FTIME 60   120   180   240   300   360   420   480   540   1800
FMASS 0 0.00829876 0.0145228 0.0141079 0.00829876 0.00622407 0.00414938 0.00414938 0.0033195 0.00311203 0.00414938
FQDOT 0 200000. 350000. 340000. 200000. 150000. 100000. 100000. 80000. 75000. 100000.
HCR   0.16  0.16  0.16  0.16  0.16  0.16  0.16  0.16  0.16  0.16  0.16
!Burn another object, still in compartment 1
OBJECT WARDROBE  1   0   1   2.15   1.65   0.00
!Add a target - now there are three
TARGET 1  2.20   1.88   2.34   0.00   0.00   1.00 CONCRETE IMPLICIT PDE
```

All of the inputs to the model are discussed in this chapter. Following the discussion that details each input, their engineering units and default values, notes are included that provided additional

guidance or frequently addressed problems that may be encountered for the input. These notes take the form of a bulleted list such as:

- The inputs may be integers (a simulation time of 1800 s), real numbers (a mass loss rate of 0.00829876 kg), or text (a floor material of CONCRETE), as appropriate. The input file is an ASCII text file and may be edited with any text editor. It is possible to use a word processor but it is important to save the file in ASCII text format and not in a word processing format. It is worthy to note that some word processors will save punctuation and other characters incorrectly for the simple ASCII text file used by CFAST. Use of a simple text editor is often preferred.

- Each line of input consists of a label followed by one or more alphanumeric parameters associated with that input label. The label must always begin in the first space of the line and be in capital letters. Following the label, the values may start in any column, and all values must be separated by either a comma or a space. Values may contain decimal points if needed or desired. They are not required.

- Inputs are in standard SI units. Most parameters have default values which can be utilized by omitting the keyword line. These are indicated in the discussion. The maximum line length is 1024 characters, so all data for each keyword must fit in this number of characters.

4.2 Preliminaries

4.2.1 Naming The Calculation

The first thing to do when setting up an input file is to give the simulation a name. The first line in the CFAST input data file must be the version identification along with an optional short description for the simulation. This is a required input. The VERSN command is the line that CFAST keys on to determine whether it has a correct data file. The format is fixed - the data must appear in the columns specified. There is no default.

Table 1. The VERSN command (simulation title)

```
VERSN Version_Number Title

Example: VERSN   5 CFAST Simulation
```

Version Number: The version number parameter specifies the version of the CFAST model for which the input data file is prepared. Normally, this is 5, although the program can make use of input files for earlier versions of the model. The number must be in column 9.
Title: The title is optional and may consist of letters, numbers, and/or symbols that start in column 11 and may be up to 50 characters. It permits the user to label each run.

Example:

```
VERSN   5 Cable Tray Fire - Base Case
```

4.2.2 Setting Time Limits and Output Options

A TIMES line specifies the length of time over which the simulation takes place and how often output will be generated. This is a required input. There are one to four entries in this line.

Table 2. The TIMES command (simulation and output times)

```
TIMES Simulation_Time Print_Interval History_Interval Spreadsheet_Interval

Example: TIMES 1800    50    0    10
```

Parameter	Units	Default
Simulation_Time: The length of time over which the simulation takes place. This is a required input which should be entered even if all other fields are not included. The maximum value for this input is 86400 s (1 day).	s	0
Print_Interval: The print interval is the time interval between each printing of the output values. If omitted or less than or equal to zero, no output values will occur.	s	0

```
TIMES Simulation_Time Print_Interval History_Interval Spreadsheet_Interval

Example: TIMES 1800    50    0    10
```

History_Interval: CFAST can store all of the results of the model simulation in a binary-formatted file which can be saved for later analysis or output using utility programs that come with the CFAST. software. This input defines the time interval between output of the model results in binary format. If a non-zero value is specified, the name of the output file for the binary-formatted history must also be specified using the DUMPR keyword described on page 16.	s	0
Spreadsheet_Interval: CFAST can output a subset of the results of the model simulation in a comma-delimited alphanumeric format which can be read by most spreadsheet software. This is designed to be imported into a spreadsheet for further analysis or graphing of the results of the simulation. This input defines the time interval between output of the model results in a spreadsheet-compatible format. A value greater than zero must be used if the spreadsheet file is to be used.		

Example:

```
TIMES    200    10    5    2
```

would specify a simulation duration of 200 s, with a text output every 10 s, an entry written to the binary history file every 5 s, and output to a spreadsheet-compatible file every 2 s.

4.2.3 Input and Output Files

In addition to the input data file created specifically for a CFAST simulation, there are several files which CFAST uses to define default values and other input information and to output the results of the simulation for later analysis. They include 1) a thermal properties file, 2) a file of predefined fire objects, 3) a binary history file, 4) a spreadsheet-compatible output file, and 5) a restart file.

The thermal properties file contains material properties for compartment surfaces, target objects that may be placed in compartments in the simulation to monitor surface temperature and heat flux to the objects, and fire objects in addition to the main fire in the simulation that may ignite based on their surface temperature or incident flux onto the surface of the object. The predefined fire objects file contains definition for a number of reference fires from the literature that may be

14

included in a simulation. The thermal properties and fire objects files may be modified by the user. Details of the files are included in the section on databases. There are default files included in the CFAST distribution if these keywords are not included in an input data file.

Table 3. The OBJFL command (object database file specification)

OBJFL Object_Fire_File Example: OBJFL objects.df	
Parameter	Default
Object_Fire_File: The name specifies a file from which the program reads fire object data.	objects.df

- There is a default version distributed with CFAST entitled objects.df. It is an indexed database. The items in the file are detailed in the report by Gross [8].

- The index is objects.ndx. If the index does not exist, CFAST will create it. If the database is modified, this index must be deleted, to force the model to reindex it. Otherwise, the input/output routines will not be able to make a correct association between the object name and the object data. The format of this database is detailed in Appendix A.

Table 4. The THRMF command (thermophysical properties database file specification)

THRMF Thermophysical_Properties_File Example: TMRMF Thermal.df	
Parameter	Default
Thermophysical_Properties_File: The name specifies a file from which the program reads data for names specified in CEILI/WALL/FLOOR. The default name is THERMAL.DF which is included with the CFAST distribution. The format of this file is detailed in Appendix A. As with the objects database, it is an indexed file, the index file name being THERMAL.NDX. If the database is modified, then the index must be deleted to force the model to re-index the file and ensure that the model uses correct material thermophysical properties.	thermal.df

The output of the simulation may be written to a disk file for further processing by programs such as REPORT or REPORTSS or to restart the CFAST model from the end of a previous simulation. At each interval of time as specified by the history interval in the TIMES label, the output is written to the file specified. For efficient disk storage and optimum speed, the data is stored in an internal binary format and cannot be read directly with a text editor. For particularly

complex scenarios or for debugging purposes, a restart command is included. The RESTR line is an optional line used to restart the model at a specified simulation time within an existing history file. These are optional parameters. In order to generate the output files, both these names must be specified, and the TIMES line (section 4.2.2) must contain a non-zero interval.

Table 5. The ADUMP command (spreadsheet output)

```
ADUMP Spreadsheet_File Output_Options

Example: standard.csv WINFS
```

Spreadsheet File: The name specifies a file to which the program outputs are written in a form readable by spreadsheet programs. This is an optional command. If omitted, the file will not be generated.

Output Options: specifies the type of output(s) written to the spreadsheet file. One or more of the letters WINFS may be specified. The default is IN. Additional parameters are also allowed as follows:

W – include wall surface and target fluxes in the output
I – include model initial conditions and scenario description in the output
N – include "normal" printout of layer temperatures, interface height, and fire size in the
output
F – include vent flows in the output
S – include layer and wall surface species concentrations in the output

Table 6. The DUMPR command (binary data output)

```
DUMPR History_File

Example: DUMPR standard.hi
```

History_File: The name specifies a file to which the program outputs for plotting are written. Note that in order to obtain a history of the variables, this parameter must be specified, and the history interval must be set to a non-zero value on the TIMES line (section 4.2.2).

Table 7. The RESTR command (simulation restart)

```
RESTR Restart_File Restart_Time

Example: RESTR standard.hi 120
```

Restart File: The name specifies a file from which the program reads data to restart the model. This data must have been generated (written) previously with the history parameter discussed earlier.

Restart Time: A time step is given after the name of the file and specifies at what time the restart should occur.

Example:

```
DUMPR PRM.HI
ADUMPF PRM.CSV NF
RESTR PRM.HI 120
```

The third command, for example, would read the PRM.HI file and initialize the run from the output saved in the history file at the 120 s interval of that calculation.

- It should be noted that there are two ways to generate a comma delimited file to import into a spreadsheet: either by direct specification in the data file using the ADUMP keyword, or through use of REPORTSS (see section 3.5.2), using data from a previously-generated history file.

4.2.4 Ambient Conditions

Ambient conditions define the environment at which the scenario begins. This allows the user to specify the temperature, pressure, and station elevation of the ambient atmosphere, as well as the absolute wind pressure to which the structure is subjected. Pressure interior to a structure is calculated simply as a lapse rate (related to the height above sea level) based on the NOAA/NASA tables [9]. This modification is applied to the vents which connect to the exterior ambient. The calculated pressure change is modified by the wind coefficient for each vent. This coefficient, which can vary from -1.0 to +1.0, nominally from -0.8 to +0.8, determines whether the vent is facing away from or into the wind. The pressure change is multiplied by the vent wind coefficient and added to the external ambient for each vent which is connected to the outside. There is an ambient for the interior and for the exterior of the structure. Three keywords define the ambient conditions: TAMB for the interior of the structure, EAMB for the exterior of the structure is EAMB, and WIND for the wind information.

Table 8. The EAMB and TAMB commands (ambient conditions)

```
TAMB Ambient_Temperature Ambient_Pressure Station_Elevation
EAMB Ambient_Temperature Ambient_Pressure Station_Elevation

Example: TAMB    300.    101300.    0.
```

Parameter	Units	Default
Ambient_Temperature: Initial ambient temperature inside (for TAMB) or outside (For EAMB) the structure at the station elevation.	K	288
Ambient_Pressure: Initial values for ambient atmospheric pressure inside (for TAMB) and outside (for EAMB) the structure at the station elevation. Default value is standard atmospheric pressure at sea level (0 m elevation) of 101.3 kPa. Input units are in Pa. These values define standard conditions as defined in Standard Atmosphere as noted in the Handbook of Chemistry and Physics [10]. There is a set of numerical approximations in the CFAST code which duplicate the pressure/temperature/altitude relationships in the handbook.	Pa	101300
Station_Elevation: The height where the ambient pressure and temperature were specified. This is the reference datum for calculating the density of the atmosphere as well as the temperature and pressure inside and outside of the structure as a function of height.	m	0

Example:

```
TAMB    300.  101300.       0
EAMB    300.  101300.       0
```

The example specifies the base temperature to be 300 K, the base pressure to be 101,300 Pa and a station elevation of 0 which is sea level.

The wind speed, scale height, and power law are used to calculate the wind coefficient for each vent connected to the outside. The wind velocity is specified at some reference height. The power law then provides a lapse rate for the wind speed. An assumption is that the wind speed is zero at the surface. The formula used to calculate the wind speed at the height of any vent is show below. The wind is applied to each external opening as a change in pressure outside of the vent.

Table 9. The WIND command (wind speed)

```
WIND Wind_Speed Scale_Height Power_Law_Coefficient

Example: WIND    10.    3.0    0.16
```

Parameter	Units	Default
Wind_Speed: Wind speed at the reference elevation.	m/s	0
Scale_Height: Reference height at which the reference wind speed is measured.	m	10
Power_Law_Coefficient: The power law used to calculate the wind speed as a function of height. Default value is 0.16. Using the notation that V_W, is the wind speed at the reference height H_W, and P_W is the power law, the exterior pressure is modified by $\delta_P = C_W \rho V^2$ and $V = V_W \left(H_i / H_W \right)^{P_W}$ where H_i is the position of the vent [1].	dimensionless	0.16

- In order to see the effect of wind, the corresponding parameter for the ventilation keyword must be specified. The default for the wind vector is 0, which turns off wind effects. Please see the HVENT command, below.

- The choice for the station elevation, temperature and pressure must be consistent. Outside of that limitation, the choice is arbitrary. It is often convenient to choose the base of a structure to be at zero height and then reference the height of the structure with respect to that height. The temperature and pressure must then be measured at that position. Another possible choice would be the pressure and temperature at sea level, with the structure elevations then given with respect to mean sea level. This is also acceptable, but somewhat more tedious in specifying the construction of a structure. Either of these choices works though so long as they are consistent. Usually, the station elevation is set to zero and the pressure to the current ambient. The effect of changing these values is small for small changes. There will be an effect for places at altitude such as Denver, Colorado but even there the effect is not pronounced. Note that the equations implemented in the model are not designed to handle negative elevations and altitudes. It is suggested that the defaults be used.

- These three parameters are optional. If they are not included in the input file, default values are used.

4.3 Compartment Geometry

In order to model a fire scenario, the size and elevation of each compartment in the structure must be specified. For a compartment, the width, depth, compartment height and height of the floor of the compartment provide this specification. The maximum number of compartments for version 5 is thirty. The usual assumption is that compartments are rectangular parallelepipeds. However, the CFAST model can accommodate odd shapes as equivalent floor area parallelepipeds or with a cross-sectional area that varies with height.

At least one compartment must be specified in the input file. There are no defaults for compartment size. There are defaults for absolute positioning (0,0,0). The fully mixed (single zone) and corridor models are turned off by default.
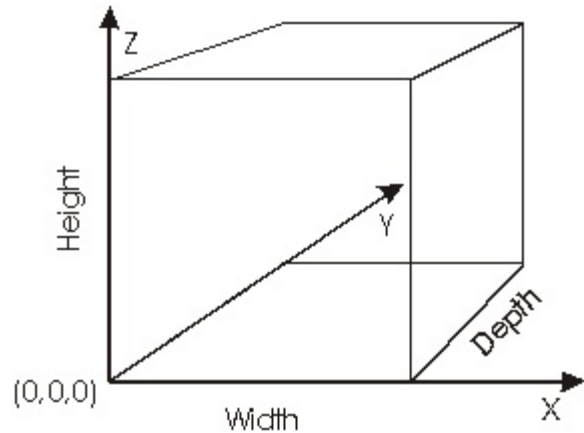
4.3.1 Defining Compartment Size and Position

Compartments in CFAST are most typically defined by a width, depth, and height. If desired, the user can prescribe the cross-sectional area of the compartment as a function of height from floor to ceiling for other shapes. The absolute position of the compartment with respect to a single structure reference point can be defined to ease visualization or to allow exact placement of vents and surfaces relative to other compartments in a detailed calculation. This specification is important for utilizing the corridor flow algorithm with the HALL command and for positioning the compartments for visualization in SMOKEVIEW. The relevant CFAST keywords are WIDTH, DEPTH, HEIGHT, CXABS, CYABS, and HI/F. This is required as a basic definition for a compartment, even if there are subsequent modifications by the ROOMA and ROOMH keywords which follow.

Table 10. The WIDTH command (compartment width)

WIDTH Width_1 Width_2 ... Width_N Example: WIDTH 10.67 10.67 1.49 1.37 1.37 10.70		
Parameter	Units	Default Value
Width1, Width_2, ... Width_N: specifies the width of each of the compartments as measured on the X axis from the origin (0,0,0) of the compartment. The number of values on the line must equal the number of compartments in the simulation.	m	none

Table 11. The DEPTH command (compartment depth)

DEPTH Depth_1 Depth_2 ... Depth_N Example:DEPTH 8.35 6.58 14.94 14.94 14.90 12.10		
Parameter	Units	Default Value
Depth_1 Depth_2 ... Depth_N: specifies the depth of each of the compartments as measured on the Y axis from the origin (0,0,0) of the compartment. The number of values on the line must equal the number of compartments in the simulation.	m	none

Table 12. The HEIGH command (compartment height)

HEIGH Height_1 Height_2 ... Height_N Example: HEIGH 3.05 3.05 3.05 3.05 3.05 3.05		
Parameter	Units	Default Value
Height_1 Height_2 ... Height_N: specifies the height of each of the compartments as measured on the Z axis from the origin (0,0,0) of the compartment. The number of values on the line must equal the number of compartments in the simulation.	m	none

Table 13. The HI/F command (floor height)

HI/F Floor_Height_1 Floor_Height_2 ... Floor_Height_N Example: HI/F 0.00 0.00 0.00 0.00 0.00 0.00		
Parameter	Units	Default Value
Floor_Height_1 Floor_Height_2 ... Floor_Height_N: specifies the height of the floor of each compartment with respect to station elevation specified by the TAMB parameter. The reference point must be the same for all elevations in the input data. The number of values on the line must equal the number of compartments in the simulation.	m	0

Table 14. The CXABS and CYABS commands (compartment position)

CXABS Absolute_X_1 Absolute_X_2 ... Absolute_X_N CYABS Absolute_X_1 Absolute_X_2 ... Absolute_X_N Example: CXABS 0. 0. 2.4 2.4 3.6 3.6		
Parameter	Units	Default Value
Absolute_X_1 Absolute_X_2 ... Absolute_X_N: specify the absolute x, y coordinates of the lower, left, back corner of the room. The number of values on the line must equal the number of compartments in the simulation.	m	0

Example:

```
WIDTH    4.00    2.00
DEPTH    4.00    5.00
HEIGH    2.30    2.30
HI/F     0.00    2.30
CYABS    0.00    2.00
```

The above example configures the model for two compartments. The first is 4.0 m x 4.0 m x 2.3 m. The second is 2.0 m x 5 m x 2.3 m. The first
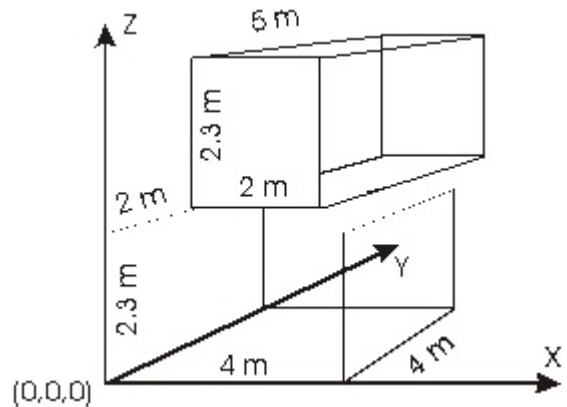
compartment is at the station elevation and the second is 2.3 m above the first. The configuration would be used, for example, to build a two floor structure. The position of the front, lower corner of compartment 2 is at (0.0 m, 2.0 m, 2.3 m).

Example:

An example from the file TEST03.DAT for a 6 compartment structure:

```
HI/F     0.00    0.00    0.00    0.00    0.00    0.00
WIDTH   10.67   10.67    1.49    1.37    1.37   10.70
DEPTH    8.35    6.58   14.94   14.94   14.90   12.10
HEIGH    3.05    3.05    3.05    3.05    3.05    3.05
```

The code defines six compartment, all at the station elevation.

4.3.2 Modeling the Compartment as a Tall Shaft or Long Corridor

For tall compartments or those removed from the room of fire origin, the compartment may be modeled as a single, well-mixed zone rather than the default two-zone assumption. A single zone approximation is appropriate for smoke flow far from a fire source where the two-zone layer stratification is less pronounced than in compartments near the fire. This is used in situations where the stratification does not occur. Examples are elevators, shafts, complex stairwells, and compartments far from the fire.

Table 15. The SHAFT command (tall compartments)

SHAFT Compartment Example: SHAFT 3		
Parameter	Units	Default Value
Compartment: Compartment to be modeled as a single, well-mixed zone	none	OFF

Example:

```
SHAFT 1
```

would use the single zone approximation for compartment 1.

For long hallways or corridors, there can be a significant delay time for the initial hot gas layer to travel along the ceiling to the far end of the compartment. By estimating the time required for a ceiling jet to travel in a corridor and the temperature distribution along the corridor, CFAST can delay flow into compartments connected to corridors until the ceiling jet has passed the connections to these compartments.

Table 16. The HALL command (long compartments)

```
HALL Compartment Velocity Depth Decay_Distance

Example: HALL   3    0.15    0.1    16.7
```

Parameter	Units	Default Value
Compartment: Compartment to be modeled as a hallway	none	none
Velocity: Optional parameter, ceiling jet velocity at the distance from the reference point where the temperature falls off by 50 %.	m/s	See note
Depth: Optional parameter, ceiling jet depth at the distance from the reference point where the temperature falls off by 50 %.	m	See note
Decay Distance: Optional parameter, distance from the reference point where the temperature falls off by 50 %.	m	16.7

Note: Default velocity and depth are calculated by the program based on the vent size, flow and layer thickness in the horizontal vent from which flow first enters the hallway, and on the width of the hallway [11].

Example:

```
HALL 1
```

would use the corridor flow model to model compartment number one as a long corridor.

- Since the algorithm depends on the flow into the hallway to determine default ceiling jet velocity and depth, the main fire compartment should not be specified as a hallway. In the main fire compartment, a stable layer will build before flow into the compartment through horizontal vents.

- The zone model concept best applies for an enclosure in which the width and length are not too different. If the horizontal dimensions of the room differ too much (i.e., the room looks like a corridor), the flow pattern in the room may become asymmetrical. If the enclosure is too shallow, the temperature may have significant radial differences. The width of the plume may at some height become equal to the width of the room and the model assumptions may fail in a tall and narrow enclosure. Therefore, the user should

24

recognize approximate limits on the ratio of the length (L), width (W), and height (H) of the compartment.

If the aspect ratio (length/width) is over about 10, the corridor flow algorithm should be used. This provides the appropriate filling time. Similarly, for tall shafts (elevators and stairways), a single zone approximation is more appropriate and the shaft option should be used. It was found experimentally [12] that the mixing between a plume and lower layer due to the interaction with the walls of the shaft, caused complete mixing. The is the flip side of the corridor problem and occurs at a ratio of the height to characteristic floor length of about 10. The following quantitative limits are recommended:

Table 17. Recommended compartment dimension limits

Group	Acceptable	Special consideration required	Corridor flow or shaft algorithm
$(L/W)_{max}$	L/W<3	3<L/W<5	L/W>5
$(L/H)_{max}$	L/H<3	3<L/H<6	L/H>6
$(W/H)_{min}$	W/H>0.4	0.2<W/H<0.4	W/H<0.2

4.4 Properties of Bounding Surfaces

To calculate heat loss through the ceiling, walls, and floor of a compartment, the properties of the bounding surfaces must be known. This includes the thermophysical properties of the surfaces and the arrangement of adjacent compartments if calculation of intercompartment heat transfer is to be calculated.

4.4.1 Thermal Properties of Boundaries

The thermophysical properties of the surfaces which define compartments are described by specifying the thermal conductivity, specific heat, emissivity, density, and thickness of the enclosing surfaces for each material and then assigning the material to the ceiling, walls, and floor of a compartment. Currently, thermal properties for materials are read from a thermal data file unique to CFAST. The thermophysical properties are specified at *one* condition of temperature, humidity, etc. There can be as many as three layers per boundary, but they are specified in the thermal database itself. (See the explanation in the section on auxiliary files for additional details.)

The default name for the thermal properties database is thermal.df. Another name can be used using the keyword THRMF in the input data file for the scenario of interest. This thermal data file is also used for targets and other objects.

4.4.2 Assigning Boundaries to Compartments

The bounding surfaces are the ceilings, walls and floors that define a compartment. These are referred to as thermophysical boundaries, since each participates in conduction and radiation as well as defining the compartments, unless these phenomena are explicitly turned off. The relevant keywords are CEILI for compartment ceilings, WALLS for compartment walls, and FLOOR for compartment floors. The input on these lines must have the same number of entries as the compartment definitions set by HI/F, HEIGH, DEPTH and WIDTH keywords.

Table 18. The CEILI command (ceiling materials)

```
CEILI Ceiling_1 Ceiling_2 ... Ceiling_N

Example:CEILI    GYPSUM    GYPSUM
```

Ceiling_1 Ceiling_2 ... Ceiling_N: The label CEILI indicates that the names of thermo-physical properties on this line describe the ceiling material. If this parameter is present, there must be an entry for each compartment.

Table 19. The WALLS command (wall materials)

```
WALLS Wall_1 Wall 2 ... Wall_N

Example: WALLS    GYPSUM    GYPSUM
```

Wall_1 Wall 2 ... Wall_N: The label WALLS indicates that the names of thermophysical properties on this line describe the wall material. If this parameter is present, there must be an entry for each compartment.

Table 20. The FLOOR command (floor materials)

```
FLOOR Floor_1 Floor_2 ... Floor_N

Example: FLOOR    CONCRETE    PLYWOOD
```

Floor_1 Floor_2 ... Floor_N: The label FLOOR indicates that the names of thermophysical properties on this line describe the floor material. If this parameter is present, there must be an entry for each compartment.

Example:

```
CEILI GYPSUM    GYPSUM
WALLS PINEWOOD  PINEWOOD
FLOOR CONCRETE  CONCRETE
```

would assign properties of GYPSUM

```
GYPSUM   0.16 900 790 0.016 0.9 0.0063 191.8 0.0587 7476 193 1.021 0.431  Gypsum Board (5/8")
```

to the ceiling of compartments one and two. Similarly for WALLS and FLOOR. In order to set the parameters for only some compartment surfaces, the entry OFF will turn off the convection/conduction for that surface for that compartment. For example:

```
CEILI GYPSUM    OFF
WALLS PINEWOOD  PINEWOOD
FLOOR CONCRETE  CONCRETE
```

would turn off the convection/conduction for the ceiling in the second compartment, leaving everything else the same.

- If the thermophysical properties of the enclosing surfaces are not included, CFAST will treat them as adiabatic (no heat transfer).

- If a name is used which is not in the database, the model should stop with an error message. The keyword in the data file simply gives a name (such as CONCRETE) which refers to the properties for that material in the thermal data file (see section 4.2.3 and Appendix A for details on the thermophysical database).

- Since most of the heat conduction is through the ceiling, and since the conduction calculation takes a significant fraction of the computation time, it is recommended that initial calculations be made using the ceiling only. Adding the walls generally has a small effect on the results, and the floor contribution is usually negligible. Clearly, there are cases where the above generalization does not hold, but it may prove to be a useful screening technique. A caveat in including floor properties is that the set of equations describing heat transfer becomes difficult to solve once the thermal wave from the compartments reaches the unexposed side of a floor. The back surfaces of compartments are assumed to be exposed to ambient conditions unless specifically specified (see section 4.7.4 to specify heat transfer connections between compartments).

4.5 Compartment Connections

Flow through vents can be natural flow through doors, windows, or openings in ceilings and floors; or forced flow in a mechanical ventilation system. Natural flow comes in two varieties. The first is referred to as horizontal flow. It is the flow which is normally thought of in discussing fires. It encompasses flow through doors, windows and so on. The other is vertical flow and can occur if there is a hole in the ceiling or floor of a compartment. This latter phenomena is useful in some scenarios such as in a ship where openings in floors and ceilings through scuttles are common and in buildings with manual or automatic heat and smoke venting.

Flow through normal vents is governed by the pressure difference across a vent. There are two situations which give rise to flow through vents. The first is flow of air or smoke driven from a compartment by buoyancy. The second type of flow is due to expansion which is particularly important when conditions in the fire environment are changing rapidly. Rather than depending entirely on density differences between the two gases, the flow is forced by volumetric expansion.

In addition to natural flow, forced flow from mechanical ventilation can affect a fire as well. More important than affecting the fire, however, is the dispersal of the smoke and toxic gases from the fire to adjacent spaces if ventilation is left operating after a fire starts.

Atmospheric pressure is about 100 000 Pa. Fires produce pressure changes from 1 Pa to 1000 Pa and mechanical ventilation systems typically involve pressure differentials of about 1 Pa to 100 Pa. In order to address pressure-induced flow correctly, pressure differences of about 0.1 Pa out of 100 000 Pa for the overall problem, or 10^{-4} Pa for adjacent compartments must be tracked.

The keywords which describe the various flow regimes are

Windows and doors (horizontal flow through vertical vents), section 4.5.1:

HVENT	specify vent which connect compartments horizontally
CVENT	opening/closing parameter

Holes in a ceiling/floor (vertical flow through horizontal vents), section 4.5.2:

VVENT	specify a vent which connects compartments vertically

HVAC specification, section 4.5.3:

MVDCT	describe a piece of (circular) duct work
MVFAN	give the pressure - flow relationship for a fan
MVOPN	Specify an opening between a compartment and ventilation system

Each of these commands is discussed in the sections that follow.

4.5.1 Defining Horizontal Natural Flow Connections (Doors, Windows)

Horizontal flow connections may include doors between compartments or to the outdoors as well as windows in the compartments. These specifications do **not** necessarily correspond to physically connecting the walls between specified compartments. Rather, lack of an opening simply prevents flow between the compartments. Horizontal flow connections may also be used to account for leakage between compartments or to the outdoors.

Horizontal connections can only be created between compartments that physically overlap in elevation at some point. These may include doors between compartments or windows in the compartments (between compartments or to the outdoors). Openings to the outside are included as openings to a compartment with a number one greater than the number of compartments described in the geometry section. The relevant keyword is HVENT. If the HVENT line is entered, the first six entries on the line are required. There is an optional seventh parameter to specify a wind coefficient.

Table 21. The HVENT command (doors and windows)

HVENT First_Compartment Second_Compartment Vent_Number Width Soffit Sill Wind First_Compartment_Position Second_Compartment_Position Example: HVENT 1 2 1 2.4 1.00 0 0 0 0		
Parameter	Units	Default
First_Compartment: First of the two compartments to be connected by a horizontal flow vent. An entry is required for each vent.		None
Second_Compartment: Second of the two compartments to be connected by a horizontal flow vent. An entry is required for each vent.		
Vent_Number: It is possible to define a total of four horizontal flow connections between any pair of compartments. A number from 1 to 4 uniquely identifies the connection. The number can be chosen as desired by the user. An entry is required for each vent.		
Width: The width of the opening. An entry is required for each vent	m	
Soffit: Position of the top of the opening relative to the floor of the compartment selected as the first compartment. An entry is required for each vent. .	m	

```
HVENT First_Compartment Second_Compartment Vent_Number Width Soffit Sill
Wind First_Compartment_Position Second_Compartment_Position

Example: HVENT   1  2  1   2.4    1.00   0      0     0   0
```

Sill: Sill height is the height of the bottom of the opening relative to the floor of the compartment selected as the first compartment. An entry is required for each vent.	m	
Wind: The wind coefficient is the cosine of the angle between the wind vector and the vent opening. This applies only to vents which connect to the outside. The range of values is -1.0 to 1.0 with a default value of zero. This is an optional parameter.	none	0
There are two additional parameters which are used to locate the compartments relative to each other. These are used to incorporate additional three dimensional information of the relative location of the vents with respect to each other. In the compartment view of CFAST, the orientation is that the rotation/translation point of the compartment is the back/bottom/left. In this view, both parameters would be with respect to the left hand side of the respective compartments. This allows the corridor filling model to incorporate a delay time for filling based on the separation between the vents. These parameters are needed only if the HALL command is used. **First_Compartment_Position:** Horizontal distance between the centerline of this vent and the reference point in the first compartment.	m	0
Second_Compartment_Position: Horizontal distance between the centerline of this vent and the reference point in the second compartment.	m	0

Examples:

```
HVENT   1  2  1   2.4    1.00   0      0    0   0
HVENT   2  3  3   1.07   2.00   1.00   0.00
```

The first example defines a door from compartment 1 to compartment 2, which has a width of 2.4 m, a soffit at 1.0 m above the floor and a sill at the floor. The wind is turned off and there is no offset for the vent for either the first or second compartment.

The second example defines Vent 3 from compartment 2 to compartment 3. The width is 1.07 m, the soffit at 2.0 m and the sill at 1.0 m. This is a large window.

- The soffit and sill specifications are with respect to the first compartment specified and is not necessarily symmetric since the elevation of the second compartment may be different than the first. Reversing the order of the compartment designations does make a difference.

Controlling Horizontal flow

Horizontal flow vents may be opened or closed during the fire with the use of the CVENT keyword. The initial format of CVENT is similar to HVENT specifying the connecting compartments and vent number. Each CVENT line in the input file details the open/close time dependent characteristics for one horizontal flow vent by specifying a fractional value as a function of time. The default is 1.0 which is a fully open vent. A value of 0.5 would specify a vent which was halfway open. If this keyword is not present, then the vent will be open for the duration of the fire. The time line for opening and closing vents derives from the time history of the fire, so the number of entries on the line beyond the specification of the vent MUST match the number of entries describing the evolution of the main fire. In order to use this keyword, at least FQTIME must be specified (see the properties of the main fire).

Table 22. The CVENT command (opening and closing vents)

```
CVENT First_Compartment Second_Compartment Vent_Number Width_1 Width_2 ...
Width_N

Example: CVENT  1  3  1   1.00   0.500
```

Parameter	Units	Default
First_Compartment: First of the two compartments to be connected by a horizontal flow vent. This must correspond to an existing horizontal flow vent	none	none
Second_Compartment: The second compartment is the compartment number to which the first compartment is connected. This must correspond to an existing horizontal flow vent	none	
Vent_Number: It is possible to define a total of four horizontal flow connections between any pair of compartments. A number from 1 to 4 uniquely identifies the connection. The number can be chosen as desired by the user. This must correspond to an existing horizontal flow vent	none	
Width_1 Width_2 ... Width_N: Fraction of the full-open width that the vent is open. This applies to the width only. The sill and soffit are not changed. The number of values on the line must equal the number of points on the fire timeline (FQTIME+1).	non-dimensional	1

Example:

```
CVENT   1  3  1   1.00   0.500
CVENT   2  3  1   1.00   0.500
```

This code would set the vent open at the beginning of the simulation (time is 0.0) and half-open at the second time node. As with the fire inputs, there is a linear interpolation from node to node.

4.5.2 Defining Vertical Natural Flow Connections (Ceiling or Floor Holes, Hatches)

This section of the input data file describes these vertical flow openings. Examples of these openings are scuddles in a ship, or a hole in the roof of a residence. Combined buoyancy- and pressure-driven (i.e., forced) flow through a vertical flow vent is possible when the connected spaces the vent are filled with gases of different density in an unstable configuration, with the density of the top space larger than that of the bottom space. With a moderate cross-vent pressure difference, the instability leads to a bi-directional flow between the two spaces. For relatively large cross-vent pressure difference the flow through the vent is unidirectional, from the high- to the low-pressure space.

Connections can exist between compartments or between a compartment and the outdoors. Openings to the outside are included as openings to a compartment with a number one greater than the number of compartments defined in the scenario (see section 4.3.1). These connections are described by the VVENT keyword. Each VVENT line in the input file describes one vertical vent. There are four parameters, the connected compartments, the shape of the opening, and the effective area of the vent.

Table 23. The VVENT command (ceiling or floor holes, hatches)

```
VVENT Top_Compartment Bottom_Compartment Area Shape

Example: VVENT   2   1  1.00   1
```

Parameter	Units	Default
Top_Compartment: The first compartment or compartment on top. The vent is through the floor of this compartment.	none	Required
Bottom_Compartment: The second compartment or compartment on the bottom. The vent is through the ceiling of this compartment..	none	
Area: specifies the cross-sectional area of the vent connecting the two compartments	m^2	
Shape: The shape factor is 1 for circular openings and 2 for square openings.	none	1

Example:

```
VVENT   2   1  1.00   1
```

- At the present time, there is not an equivalent CVENT mechanism for opening or closing the vertical vents.

- Vertical connections can only be created between compartments that could be physically stacked based on specified floor and ceiling elevations for the compartments. Some overlap between the absolute floor height of one compartment and the absolute ceiling height of another compartment is allowed. However, whether the compartments are stacked or overlap somewhat, the ceiling/floor absolute elevations must be within 0.01 m of each other. The check is not done when the connection is to the outside.

4.5.3 Defining Mechanical Ventilation Connections

Fan-duct systems are commonly used in buildings for heating, ventilation, air conditioning, pressurization, and exhaust. Generally, systems that maintain comfortable conditions have either one or two fans. Residences often have a systems with a single fan. Further information about these systems is presented in Klote and Milke [13] and the American Society of Heating, Refrigerating and Air Conditioning Engineers [14].

The model for mechanical ventilation used in CFAST is based on the theory of networks and is based on the model developed by Klote [15]. This is a simplified form of Kirchoff's law which

says that flow into a node must be balanced by flow out of the node. The equations used describe the relationship between the pressure drop across a duct, the resistance of a duct, and the mass flow. The pressure can be changed by conditions in a compartment, or a fan in line in the duct system. Resistance arises from the finite size of ducts, roughness on duct surfaces, bends and joints.

Each fan-duct system is represented as a network of nodes, each at a specific temperature and pressure. The nodes may be connected by fans, ducts, fittings and other components. Except for fans, air flows through these components from nodes of higher pressure to nodes of lower pressure.

The MVOPN line is used to connect a compartment to a node in the mechanical ventilation system. The elevation for each of these exterior nodes is specified as a relative height to the compartment floor on the MVOPN line. The MVDCT keyword is used to specify a piece of the mechanical ventilation duct work. All node elevations can be specified, but elevations for the exterior nodes, that is those connected to a compartment, are ignored. These heights are determined by entries on the MVOPN line. The heights for interior nodes are absolute heights above the reference datum specified by TAMB. The heights are specified in pairs with the node number followed by the height.

Table 24. The MVDCT command (mechanical ventilation ductwork)

```
MVDCT First_Node Second_Node Length Diameter Absolute_Roughness
First_Flow_Coefficient First_Area, Second_Flow_Coefficient Second_Area

Example: MVDCT   1   2   2.30   0.10 .00200   0.00 1.0000   0.00 1.0000
```

Parameter	Units	Default
First_Node: First node number. This is a node in the mechanical ventilation scheme, not a compartment number (see MVOPN).	none	none
Second_Node: Second node number.	none	none
Length: Length of the duct.	m	none
Diameter: Equivalent diameter of the duct. All duct work is assumed to be circular. Other shapes must use an equivalent diameter and an appropriate flow coefficient (see for example [16] and [17])	m	none
Absolute_Roughness: The absolute roughness affects laminar flow friction losses within the duct. It is obtained from reference books and undergraduate texts on fluid flow [18].	m	0.0

```
MVDCT First_Node Second_Node Length Diameter Absolute_Roughness
First_Flow_Coefficient First_Area, Second_Flow_Coefficient Second_Area

Example: MVDCT   1   2   2.30   0.10 .00200   0.00 1.0000   0.00 1.0000
```

First_Flow_Coefficient: Flow coefficient to allow for an expansion or contraction at the end of the duct which is connected to node number one. To use a straight through connection (no expansion or contraction) set to zero. Values for these flow coefficients have been tabulated for a substantial number of connection geometries [18], [19].	none	1.0
First_Area: Cross-sectional Area of the duct at the first node.	m^2	none
Second_Flow_Coefficient: Coefficient for second node.	none	1.0
Second_Area: Cross-sectional area at the second node.	m^2	none

- All duct work is assumed to be circular. For square ducts, use an effective radius. The roughness should be a small fraction of the duct diameter. This is the case in properly installed mechanical ventilation systems and the solution of the equations assumes this.

- Examples of roughness values found in typical duct work are available [20]

Table 25. Typical values for duct roughness

Duct Material	Category	Absolute Roughness (m)
Uncoated Carbon Steel, Clean. PVC Plastic Pipe	Smooth	0.00003
Galvanized Steel, Longitudinal Seams	Medium	0.00009
Galvanized Steel, Longitudinal Seams	Average	0.00015
Fibrous Glass Duct	Rough	0.0009
Fibrous Glass Duct Liner, Flexible Duct, Metallic. Flexible Duct, All Types of Fabric and Wire	Rough	0.003

- The equations describe mechanical ventilation through a series of complex non-linear algebraic equations. As such, they are difficult to solve and there are several errors which might be encountered in using this system. Generally, the problem will manifest itself

with an error returned from the solver indicating that a solution could not be reached. In the section on error codes, these usually are returned as negative error codes.

- Nodes specified by each MVDCT entry must connect with other nodes, fans, or compartments. Do not specify ducts which are isolated from the rest of the system.

- Nodes specified by each MVDCT entry must connect with other nodes, fans, or compartments. Do not specify ducts which are isolated from the rest of the system.

- Elevations of the interior mechanical ventilation nodes can be done using the INELV keyword. The specification of the interior elevations (INELEV) is maintained for completeness with the validated ASHRAE model of mechanical ventilation [14]. It accounts for the change of potential energy ($\rho g h$) of mass moving from the top of a building to the bottom. However, it has little effect in buildings subjected to fires, and specifying node points outside of the building causes incorrect results. It is recommended that this keyword not be used, and a description of the height z is not included here.

Mechanical ventilation systems are connected to compartments with diffusers located in the walls or ceilings of a compartment. The relevant CFAST keyword is MVOPN.

Duct Work Node Number: The mechanical ventilation system node number to which the mechanical ventilation system diffuser is connected.

Orientation: The orientation of the diffuser relative to the floor of the compartment. A horizontal diffuser implies vertical flow through the ceiling or floor of the compartment. A vertical diffuser implies horizontal flow through a wall of the compartment.

Height: Height of the duct opening above the floor of the compartment measured from the midpoint of the register.

Table 26. The MVOPN command (mechanical ventilation compartment connections)

```
MVOPN Compartment_Number Duct_Work_Node Orientation Height Area

Example: MVOPN   1   1 V   2.10   0.12
```

Parameter	Units	Default
Compartment Number: The compartment to which the mechanical ventilation system diffuser is connected	Integer	none
Duct Work Node Number: The mechanical ventilation system node number to which the mechanical ventilation system diffuser is connected.	Integer	none
Orientation: The orientation of the diffuser relative to the floor of the compartment. A horizontal diffuser implies vertical flow through the ceiling or floor of the compartment. A vertical diffuser implies horizontal flow through a wall of the compartment.	V or H	V
Height: Height of the duct opening above the floor of the compartment measured from the midpoint of the register.	m	0
Area: Cross-sectional area of the opening into the compartment. The area will be truncated if the midpoint (height) is set such that the height plus or minus the effective length is above the compartment ceiling or below the floor.	m^2	0

A fan is defined using the MVFAN line to indicate node numbers and to specify the fan curve with power law coefficients.

Table 27. The MVFAN command (mechanical ventilation fans)

MVFAN First_Node Second_Node Minimum_Pressure Maximum_Pressure Coefficients		
Parameter	Units	Default
First_Node: First node in the mechanical ventilation system to which the fan is connected	Integer	none
Second_Node: Second node to which the fan is connected	Integer	none
Minimum_Pressure: Minimum pressure for which the polynomimial fan curve is valid. Below this pressure, the flow is assumed equal to the value at the minimum pressure	Pa	0
Maximum_Pressure: Above the maximum pressure, the flow is assumed equal to the value at the maximum pressure	Pa	300
Coefficients: At least one, and a maximum of five coefficients, to specify the flow as a function of pressure. The fan coefficients are simply the coefficients of an interpolating polynomial for the flow speed as a function of the pressure across the fan housing of the form: $$v = c_0 + c_1\Delta P + c_2\Delta P^2 + c_3\Delta P^3 + c_4\Delta P^4$$ where v is the flow in m³/s and ΔP is the pressure drop in Pa.	note 1	at least one

[1] Coefficients are typically a regression to experimental data. Units are appropriate so that resulting flow is in m³/s. See example below.

Example:

```
MVOPN    1    1 V    2.10    0.12
MVOPN    2    3 V    2.10    0.12
MVDCT    1    2    2.30    0.10 .00200    0.00 1.0000    0.00 1.0000
MVFAN    2    3    0.00 300.00   0.140E+00  −0.433E−03
```

In this example, the fan flow was developed from a simple set of data points of flow as a function of pressure. A straight line through these data resulted in the fan coefficients $c_0 = 0.140$ m³/s and $c_1 = -0.433 \times 10^{-3}$ m³/s Pa so the flow is

	Pressure (Pa)	Flow (m³/s)
Minimum	0.0	0.1400
Maximum	300.0	0.0101

$$v = 0.140 - 0.433 \times 10^{-3}\ \Delta P$$

where v is the flow in m³/s and ΔP is the pressure drop in Pa.. The resulting fan specification is thus:

```
MVFAN 2 3 0.0 300.0 0.140E+00 -0.433E-03
```

- CFAST does not include provisions for backward flow through a fan. Calibrations for backward flow are not provided by fan manufacturers and the equations incorporated in CFAST do not allow for such flow. The problem is simply that in this flow regime, the fan has stalled, and will soon fail. The fan curves are steady state, and there is no convenient way to represent this flow in a transient problem.

- The description of the fan includes a minimum and maximum pressure. Below the minimum the flow is assumed to be constant. Above the maximum, the flow is assumed to stop.

4.6 Prescribing Fires

A simulated fire in CFAST is implemented as a source of fuel mass which is released at a prescribed rate (the pyrolysis rate). Energy is released by the fuel and combustion products are created as it burns. The model incorporates two distinct types of fires: **unconstrained** and **constrained**. In the unconstrained fire, the fire simply releases mass and energy at a rate prescribed by the user; no calculation or tracking of products of combustion is included. In the constrained fire, species production is calculated based on production yields prescribed by the user. In addition, for the constrained fire, the pyrolysis rate and resulting energy and species generation may be limited by the oxygen available for combustion. When sufficient oxygen is available for combustion, the heat release rate for the constrained fire is the same as for the unconstrained fire.

The model can simulate multiple fires in one or more compartments of the building. These fires are treated as totally separate entities, with no interaction of the plumes. These fires are generally referred to as "objects" and can be ignited at a prescribed time, temperature or heat flux.

CFAST does not include a pyrolysis model to predict fire growth. Rather, pyrolysis rates for each fire are prescribed by the user. While this approach does not directly account for increased pyrolysis due to radiative feedback from the flame or compartment, in theory these effects could be prescribed by the user as described in this section. In an actual fire, this is an important consideration, and the specification used should consider the experimental conditions as closely as possible.

Unconstrained Fire

An unconstrained fire releases energy based on the pyrolysis rate of the fuel. This type of fire is applicable to fuels for which there is always sufficient oxygen in the fuel for combustion to take place and when the concentrations of products of combustion are not of interest.

Constrained Fire

A constrained fire releases energy based on the pyrolysis of fuel, but may be constrained by the oxygen available for combustion depending on the compartment conditions. Complete burning will take place only where there is sufficient oxygen. When insufficient oxygen is entrained into the fire plume, unburned fuel will be transported from zone to zone until there is sufficient oxygen and a high enough temperature to support combustion. In general, CFAST uses a simple definition of a combustion reaction that includes major products of combustion for hydrocarbon fuels:

$$C_m H_n O_p N_q Cl_r + m_1(O_2 + 3.76N_2) \rightarrow m_2 CO_2 + m_3 CO + m_4 H_2O + m_5 HCl + m_6 HCN + 3.76 m_1 N_2$$
$$+ \text{ soot} + \text{total unburned hydrocarbons} + \text{heat} \tag{3}$$
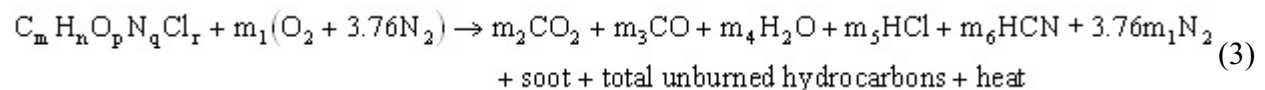
where the coefficients m_1, m_2, etc. represent appropriate molar ratios for a stoichiometric balance of the equation. For complete combustion of the simplest hydrocarbon fuel, methane reacts with oxygen to form carbon dioxide and water. The only input required is the pyrolysis rate and the heat of combustion. For fuels that contain oxygen, nitrogen, or chlorine, the reaction becomes more complex. In this case, production yields for the species are prescribed by the user. Stoichiometry is used to insure conservation of mass and elements in the reaction. The species which are calculated are oxygen, carbon dioxide, carbon monoxide, water, total unburned hydrocarbons (tuhc), and soot. Gaseous nitrogen is included, but only acts as a diluent. Production of hydrogen cyanide and hydrogen chloride are tracked solely based on user prescribed yields.

The heat release rate for a constrained fire may be reduced below its prescribed value based upon the oxygen available for combustion. For the constrained fire, the burning rate may be less than the pyrolysis rate and cannot be simplified as in the case of the unconstrained fire. As fuel and oxygen are consumed, heat is released and various products of combustion are formed.

Constraint on the Fire by Limiting Combustion by Available Oxygen (LOI)

For a constrained fire, the heat release rate is limited by available oxygen. This limit is applied in three places: The first is burning in the portion of the plume which is in the lower layer of the room of fire origin; the second is the portion of the plume in the upper layer, also in the room of origin; the third is in the vent flow which entrains air from a lower layer into an upper layer in an

40

adjacent compartment. The unburned hydrocarbons are tracked in this model. Combustion of CO to CO_2 is not included in the model. Actual combustion chemistry is not considered in CFAST due to the complexities associated with detailed kinetics and transport.

Visibility

There are two calculations involving radiation in this model. One is for energy balance and is based on broadband radiation absorption. The amount of radiation absorbed in sensitive to the species present, specifically water vapor, soot and carbon dioxide.

The other is for visibility of egress signs. This calculation is based solely on the soot concentration and is reported as optical depth (per meter). The conversion factor is based on the recent work by Mulholland and Croarkin [21]. The value for converting mass density in kg/m^3 to optical depth is 3817 m^2/kg. The value reported is intended specifically for assessing the visibility of egress signs, based on the work of Jin [22]. It is not applicable to the far blue or red regions of the spectrum and so should not be used for assessing optical detection of fires through smoke.

Since chemical species are tracked only for a constrained fire, the optical depth for an unconstrained fire is fixed at 0.5 m^{-1} for the upper layer and 0.05 m^{-1} for the lower layer.

4.6.1 Defining the Main Fire and Time-Dependent Fire Curves

The fire specifications allow the user to describe the fire source in the simulation. The location and position of the fire are specified using the LFBO, FPOS and FHIGH entries. Chemical properties of the fuel are specified with the CHEMI keyword along with miscellaneous parameters.

The time dependent variables of the fire are described by the mass loss rate, rate of heat release, fuel height, and fuel area inputs. All of these specifications are optional. The number of entries on all time dependent lines must be consistent. The count is determined by the number of entries on the FTIME line. There is an implied value of zero (beginning of the simulation) for FTIME. For every other entry (the fire parameters discussed in this section as well as the CVENT keyword) there must be one more than the values included on the FTIME line.

Species production rates are specified in a manner similar to the fire, entering the rates as a series of points with respect to time. The species which are followed by CFAST are: carbon dioxide, carbon monoxide, hydrogen cyanide, hydrogen chloride, nitrogen, oxygen, soot, total unburned hydrocarbons and water.

The keywords which set values for the primary fire are:

CHEMI   specify parameters of the combustion chemistry for the main fire
LFBO   compartment of fire origin for the main fire
LFBT   fire type (unconstrained or constrained)
FPOS   position of the fire within the compartment
FTIME   time points for the main fire curve
FMASS   pyrolysis rate of the main fire as a function of time
FQDOT   heat release rate of the main fire as a function of time
FHIGH   height of the base of the fire as a function of time
HCR   hydrogen / carbon ratio of the fuel as a function of time
CO   Carbon monoxide / Carbon dioxide ratio of the fuel as a function of time
OD   Carbon / Carbon dioxide ratio in the fuel as a function of time
O2   Oxygen / Carbon ratio in the fuel as a function of time
HCN   Hydrogen Cyanide / fuel ratio in the fuel as a function of time
HCL   Hydrochloric Acid / fuel ratio in the fuel as a function of time
CT   mass of toxic combustion products produced per mass of fuel as a function of time

For an unconstrained fire (LFBT=1), only the concentration-time product of pyrolysate (CT) can be specified. No other species are followed. For a constrained fire (LFBT=2), nitrogen (N2), oxygen (O2), carbon dioxide (CO2), carbon monoxide (CO), hydrogen cyanide (HCN), hydrogen chloride (HCL), soot (OD), unburned fuel and water (H2O) can be specified and tracked.

Fire type and location

Two CFAST keywords define the compartment of fire origin and the level of detail in the combustion chemistry calculation.

Table 29. The LFBO command (main fire location)

LFBO Compartment_of_Fire_Origin Example: LFBO 1	
Parameter	Default
Compartment of fire origin: specifies the compartment that contains the main fire for the simulation. Any compartment defined is valid. An entry of 0 turns off the main fire.	0

Table 30. The LFBT command (fire type)

LFBT Fire_Type LFBT 2	
Parameter	Default
Fire type: unconstrained, constrained, or flame spread. A value of 0 turns the main fire off, a value of 1 is the unconstrained fire, 2 the constrained fire, and 3 is an object on which fire spread (flame spread) occurs. A flame spread object can only be specified by the OBJECT keyword and there can only be one of these. For a detailed description, please see the technical reference manual [1], and in Appendix A on the "Other Objects" database.	1

Characteristics of the fuel

The CHEMIE input defines characteristics of the main fire source including heat of combustion, the fraction of heat generated by the main fire that goes into radiation, and ignition temperature of the fuel.

Table 31. The CHEMI command (main fire characteristics)

CHEMI Molar_Weight Relative_Humidity Lower_Oxygen_Limit Heat_of_Combustion Initial_Fuel_Temperature Gaseous_Ignition_Temperature Radiative_Fraction Example: CHEMI 16. 0. 10.0 18100000. 300. 400. 0.		
Parameter	Units	Default
Relative Molecular Mass: This is the conversion factor from mass density to molecular density for "Total Unburned Hydrocarbon," the fuel vapor. It is used only for conversion to parts-per-million and has no effect on the model itself.	g/mol	16
Relative humidity: The initial relative humidity in the system. This is converted to kilograms of water per cubic meter.	%	
Lower oxygen limit: The limit on the ratio of oxygen to other gases in the system below which a flame will not burn. This is applicable only to constrained or flame spread fires.	%	10

```
CHEMI Molar_Weight Relative_Humidity Lower_Oxygen_Limit Heat_of_Combustion
Initial_Fuel_Temperature Gaseous_Ignition_Temperature Radiative_Fraction

Example: CHEMI  16.    0.  10.0    18100000.  300. 400.    0.
```

Heat of combustion: The energy released per kilogram of mass burned.	J/kg	50 000 000
Initial fuel temperature: The temperature of the vapor as it is emitted from a fuel source. Typically, the initial fuel temperature is the same as the ambient temperature specified on the ambient conditions window. This may not be true if significant conductive or radiative heat transfer affects the fuel. This affects the enthalpy since temperatures below the lower layer ambient cause the plume to cool. The effect is not large, but it is noticeable.	K	Ambient
Gaseous ignition temperature: The lower temperature limit on the burning of fuel in a gas layer. Since CFAST does not support a combustion kinetics model, this is the algorithm used for fires out of vents. If omitted, the default is arbitrarily set to the initial fuel temperature plus 100 ˚C.	K	Fuel temperature plus 100 K
Radiative fraction: The fraction of the energy produced in combustion that is radiated from the fire and plume. Within CFAST, the radiative fraction defaults to 0.30 [23]; i.e., 30 % of the fire's energy is released via radiation. For other fuels, the work of Tewarson [24], McCaffrey [25], or Koseki [26] is available for reference. The typical range for the radiative fraction is from about 0.05 to 0.4.	non-dimensional	0.3

Time dependent parameters

All of the time dependent variables depend on a predefined time line. The FTIME input specifies a sequence of time points that define the timing of the main fire. An entry indicates a point on the time-line when the mass loss rate, fuel height and other time dependent values are specified for the fire. This time is independent of the simulation time which is specified by the TIMES line. If the simulation time is longer than the total duration of the fire, the final values specified for the fire (mass loss rate, fuel height, fuel area, and species) are continued until the end of the simulation.

Table 32. The FTIME command (main fire timeline)

`FTIME Fire_Time_1 Fire_Time_2 ... Fire_Time_M`
 	
`Example: FTIME  60   120   180`	

Fire_Time_1 Fire_Time_2 ... Fire_Time_M: specify a sequence of time points that define the timing of the main fire. An entry indicates a point on the time-line when the mass loss rate, fuel height and other time dependent values are specified for the fire. There is an assumed value of time equals zero for the first step, so each time dependent parameter below requires one additional entry. The model checks for consistency of the time dependent parameters, including CVENT, based on the entries of the FTIME line.	real numbers

Each of the following gives time dependent characteristics of the fire, and each of these entries is related to the values of FTIME defined above.

FHIGH	Fuel height - the height of the base of the fire (note the interaction with FPOS).
FPOS	Position of the fire within the compartment (not time dependent).
FMASS	Pyrolysis rate - The rate at which fuel is pyrolyzed at times corresponding to each time point of the specified fire.
FQDOT	Heat of combustion of the fuel.
LFBO	Compartment of origin.
HCR, O2, CO, CT, OD, HCN, HCL species yields, see definition below.	

The units for species (HCR, etc.) are kilograms per kilogram. However, the meaning is different for fuel properties and species production rates. The input for CT is the kilograms of "toxic" combustion products produced per kilogram of fuel pyrolyzed. Input for HCR is the mass ratio of hydrogen to carbon ratio in the fuel. Input for O2 is the mass ratio of oxygen to carbon *as it becomes available from the fuel*. Input the ratio of the mass of carbon to carbon dioxide produced by the oxidation of the fuel for OD. The input for CO is the ratio of the mass of carbon monoxide to carbon dioxide produced by the oxidation of the fuel.

Table 33. The FHIGH and FPOS commands (main fire position)

FHIGH Fire_Height_1 Fire_Height_2 ... Fire_Height_M Example: FHIGH 0. 0.25 0.25		
Parameter	Units	Default
Fire_Height_1 Fire_Height_2 ... Fire_Height_M: Time-based values for height of the base of the fire. Actual height of the base of the fire is the sum of the value specified by FPOS and by FHIGH for a particular time in the fire development.	m	0

FPOS Fire_Depth_Position Fire_Width_Position Fire_Height_Position Example: FPOS 1.2 1.2 0.		
Parameter	Units	Default
Fire_Depth_Position: Position of the center of the base of the fire measured in the y direction from the front lower left corner of the compartment origin (0,0,0) in the compartment of fire origin.	m	Depth/2
Fire_Width_Position: Position of the center of the base of the fire measured in the x direction from the front lower left corner of the compartment origin (0,0,0) in the compartment of fire origin.	m	Width/2
Fire_Height_Position: Position of the center of the base of the fire measured in the z direction from the front lower left corner of the compartment origin (0,0,0) in the compartment of fire origin. Actual height of the base of the fire is the sum of the value specified by FPOS and by FHIGH for a particular time in the fire development.	m	0

Table 34. The FMASS and FQDOT commands (main fire pyrolysis and heat release rates)

```
FMASS Pyrolysis_Rate_1 Pyrolysis_Rate_2 ... Pyrolysis_Rate_M

Example: FMASS 0 0.00829876 0.0145228 0.0141079
```

Parameter	Units	Default
Pyrolysis_Rate_1 Pyrolysis_Rate_2 ... Pyrolysis_Rate_M: Pyrolysis rate of the fire as a series of time points consistent with the FTIME keyword specification.	kg/s	none

```
FQDOT HRR_1 HRR_2 ... HRR_N

Example: FQDOT 0 200000. 350000. 340000.
```

Parameter	Units	Default
HRR_1 HRR_2 ... HRR_N: Heat release rate of the fire as a series of time points consistent with the FTIME keyword specification.	W	None

Table 35. The HCN, HCl, CT, HCR, O2, OD and CO commands (main fire species)

```
HCN Yield_1 Yield_2 ... Yield_M
HCL Yield_1 Yield_2 ... Yield_M
CT Yield_1 Yield_2 ... Yield_M

HCR   0.16  0.16  0.16  0.16
```

Parameter	Units	Default
Yield_1 Yield_2 ... Yield_M: Composition of the pyrolyzed fuel per mass of fuel produced in the pyrolysis of the fuel.	kg/kg	0

```
HCR H_Yield_1 H_Yield_2 ... H_Yield_M
O2 O2_Yield_1 O2_Yield_2 ... O2_Yield_M

Example: HCR    0.333    0.333    0.333
```

Parameter	Units	Default
H_Yield_1 H_Yield_2 ... H_Yield_M: the hydrogen component of the pyrolyzed fuel. HCR is the ratio of hydrogen to carbon produced in the pyrolysis of the fuel.	kg/kg	0.333
O2_Yield_1 O2_Yield_2 ... O2_Yield_M: Oxygen component of the pyrolyzed fuel. O2 is the ratio of oxygen to carbon produced in the pyrolysis of the fuel. The O2 input thus represents excess oxygen available from the fuel for combustion. For normal fuels, this input should be omitted.	kg/kg	0

```
OD Yield_1 Yield_2 ... Yield_M
CO Yield_1 Yield_2 ... Yield_M

Example: CO    0.003    0.003    0.21    0.21
```

Parameter	Units	Default
Species yields ratios of soot to carbon dioxide and carbon monoxide to carbon dioxide produced in the pyrolysis of the fuel.	kg/kg	0

Example:

```
CHEMI  16.    0.  10.0    18100000.  300. 400.   0.
LFBO   1
LFBT   2
FPOS   2.00   2.00   0.00
FTIME          400.    <<<=== There is an implicit point at time=0
FMASS  0.0014   0.0014
FHIGH  0.00     0.01
FAREA  0.00     0.00
FQDOT  2.53E+04 2.53E+04
HCR    0.333    0.333
CO     0.010    0.020
OD     0.020    0.010
```

- With the three parameters, heat of combustion (HOC) from CHEMI, FMASS and FQDOT, the pyrolysis and heat release rate are over specified. The model uses the last two of the three to obtain the third parameter. That is, if the three were specified in the order HOC, FMASS and FQDOT, then FQDOT would be divided by FMASS to obtain the HOC for each time interval. If the order were FMASS, FQDOT and HOC, then the

pyrolysis rate would be determined by dividing the heat release rate by the heat of combustion. If only two of the three are given, then those two determine the third, and finally, if none or only one of the parameters is present, default values are used.

- By default, the fire is placed in the center of the compartment on the floor. To place the fire in a different location, the FPOS keyword may be included in the input file. If values for any of the three variables are invalid (i.e., less than zero or greater than the compartment dimension in the appropriate direction), the location for that direction defaults to the center of the appropriate direction.

- It is not necessary to define a main fire. This feature can be turned off by setting the type (LFBT) to zero.

4.6.2 Prescribing Additional Burning Objects

The OBJECT keyword allows the specification of additional objects to be burned in the fire scenario. The object name and object compartment are required if the OBJECT keyword is used. All other input items have default values if they are not specified. These defaults are: start time 0.0, first element 1, depth (x position) one half the depth of the compartment, breadth (y position) one half the width of the compartment, and height (z position) 0.0. To specify any input item, all preceding items on the OBJECT line must also be specified. For example, the first element can not be set if the start time is not set. Positioning of the object within a compartment is specified in the same manner as for the main fire.

Table 36. The OBJECT command (secondary fire objects)

```
OBJECT Name Compartment Ignition_Criterion_Value Ignition_Criterion_Type
Depth_Position Breadth_Position Height_Position Normal_Dept) Normal_Breadth
Normal_Height Horizontal_Flame_Spread_Ignition_Position
Vertical_Flame_Spread_Ignition_Position

Example: OBJECT SOFA      1    10     1   4.00   2.00   0.00
```

Parameter	Units	Defaults
Name: The name from the objects database for the desired object. Specifying a name not found in the database causes CFAST to stop with an appropriate error message.	none	none
Compartment: The compartment that the object is in during the simulation. If a compartment number outside the range of specified compartments is used, CFAST provides an error message and stops.	none	none

```
OBJECT Name Compartment Ignition_Criterion_Value Ignition_Criterion_Type
Depth_Position Breadth_Position Height_Position Normal_Dept) Normal_Breadth
Normal_Height Horizontal_Flame_Spread_Ignition_Position
Vertical_Flame_Spread_Ignition_Position

Example: OBJECT SOFA       1     10      1   4.00   2.00   0.00
```

Ignition Criterion Value: The numerical value at which ignition will occur. If it is less than or equal to zero, the default is taken. For constrained and unconstrained fires, the default is and ignition at time of zero. For flame spread objects, the default is the ignition temperature in the database.	s, K, or W/m^2	0 s
Ignition Criterion Type: The type of ignition condition specified by the *Ignition Criterion Value*. Acceptable values are 1 for time, 2 for object surface temperature, and 3 for incident flux to object surface.	none	none
Depth Position: Position of the object as a distance from the rear wall of the object compartment (X direction).	m	½ depth
Breadth Position: Position of the object as a distance from the left wall of the object compartment (Y direction).	m	½ breadth
Height Position: Height of the object above the floor (Z direction).	m	0
Normal Vector (Depth): specifies a vector of unit length perpendicular to the exposed surface of the object. (Depth) component is in the direction from the rear wall of the object compartment. Default value is a horizontal, upward facing object, unit vector = (0,0,1)	m	0
Normal Vector (Breadth): specifies a vector of unit length perpendicular to the exposed surface of the object. (Breadth) component is in the direction from the left wall of the object compartment. Default value is a horizontal, upward facing object, unit vector = (0,0,1)	m	0
Normal Vector (Height): specifies a vector of unit length perpendicular to the exposed surface of the object. (Breadth) component is in the direction from the floor of the object compartment. Default value is a horizontal, upward facing object, unit vector = (0,0,1)	m	1

```
OBJECT Name Compartment Ignition_Criterion_Value Ignition_Criterion_Type
Depth_Position Breadth_Position Height_Position Normal_Dept) Normal_Breadth
Normal_Height Horizontal_Flame_Spread_Ignition_Position
Vertical_Flame_Spread_Ignition_Position

Example: OBJECT SOFA        1      10      1    4.00   2.00   0.00
```

Horizontal Flame Spread Ignition Position: Assuming the ignition surface is a wall with the corner closest to the lower back left corner of the compartment, the horizontal flame spread ignition position is the horizontal distance from the rear lower corner of the object. Only applicable to flame spread objects.	m	0
Vertical Flame Spread Ignition Position: Assuming the ignition surface is a wall with the corner closest to the lower back left corner of the compartment, the horizontal flame spread ignition position is the vertical distance from the rear lower corner of the object. Only applicable to flame spread objects.	m	0

Example:

```
OBJECT SOFA       1      10      1   4.00   2.00   0.00
OBJECT WARDROBE   1      20000   3   0.00   2.00   0.00
```

This example show two objects, a sofa and a wardrobe, both in compartment 1. The sofa is ignited at a time of 10 s in the simulation and the wardrobe is ignited when the incident flux on the surface of the wardrobe reaches 20 kW/m^2.

- Object fires are used to include additional burning objects in a simulation. Flame spread calculations are a special case of burning objects. For additional details of flame spread objects, see the CFAST technical reference guide [1], the work of Cleary and Quintiere [27], or Babrauskas and Grayson [28].

4.7 Special Features

4.7.1 Defining Targets

CFAST can track and report calculations of the net heat flux striking arbitrarily positioned and oriented targets and the temperature of these targets. Two keywords are used to specify targets:

TARGET	to calculate net heat flux to an arbitrarily-placed object in a compartment
TARG	to calculate net heat flux to an object on an interior surface of a compartment. It is a simplified version of TARGET.

In addition to any user-defined targets, there are always two targets that are automatically placed in any compartment containing a fire. Both are included for reporting purposes. The first is an "ambient target" and is intended to represent the net flux to a human body. This is used for the flux in the hazard calculation for tenability. The assumption is that the target will remain at ambient temperature, which is a surrogate for body temperature. The second determines the net flux to a horizontal target on the floor whose temperature is assumed to be the same as the floor surface. The calculation can be used to estimate the ignition of combustibles on the floor as a surrogate for estimating time time to flashover, typically taken to be 20 kW/m^2. Thus if one of the target keywords is used, it will be in addition to these two predefined targets.

For all targets, heat flux is calculated to the surface specified by the user (with the normal vector). Conduction into the target is assumed to occur only from this surface.

Table 37. The TARGET command (arbitrarily-placed target objects)

```
TARGET Compartment Depth Breadth Height Normal_Depth Normal_Breadth
Normal_Height) Material Method Equation_Type

Example: TARGET   1    7.62    1.52    4.26    0.    1.    0.
```

Parameter	Units	Default
Compartment: The compartment in which the target is located	none	none
Depth: Position of the target as a distance from the rear wall of the target compartment (X direction).	m	½ depth
Breadth: Position of the target as a distance from the left wall of the target compartment (Y direction).	m	½ breadth
Height: Height of the target above the floor (Z direction).	m	0
Normal Vector (Depth): specifies a vector of unit length perpendicular to the exposed surface of the target. (Depth) component in the direction from the rear wall of the target compartment. Default value is a horizontal, upward facing target, unit vector = (0,0,1)	none	0.0
Normal Vector (Breadth): specifies a vector of unit length perpendicular to the exposed surface of the target. (Breadth) component in the direction from the left wall of the target compartment. Default value is a horizontal, upward facing target, unit vector = (0,0,1)	none	0.0

```
TARGET Compartment Depth Breadth Height Normal_Depth Normal_Breadth
Normal_Height) Material Method Equation_Type

Example: TARGET    1    7.62    1.52    4.26    0.    1.    0.
```

Normal Vector (Height): specifies a vector of unit length perpendicular to the exposed surface of the target. (Breadth) component in the direction from the floor of the target compartment. Default value is a horizontal, upward facing target, unit vector = (0,0,1)	none	1.0
Material: An optional parameter used to specify the wall material of the target. Any material from the thermal database used to represent wall materials may be used here. Since the transient heat conduction problem is not solved now for the target this parameter is not used.	none	none
Method: Optional parameter that indicates the solution method. STEADY for steady state solution, XPLICIT for explicit solution, MPLICIT for implicit solution	none	MPLICIT
Equation Type: If METHOD is not STEADY, this parameter further indicates the equation type to be either ODE or PDE.	none	PDE

- The TARGET (and TARG) keywords perform a heat transfer calculation between the compartment and the target. The steady state option assumes that the target material reacts instantly to changing conditions and computes the target temperature that would result in a balance of incoming and outgoing heat (i.e., a steady state). If a transient target temperature is modeled, then one can either assume that there is a temperature variation within the target or assume that the target is "thin" and can be modeled using only one temperature. If the target is assumed to be thin then the ODE option should be used set since this is how the equations are solved. If the target is assumed to be thick then the PDE option should set. Finally, if one of the two transient options are set (ODE or PDE), the numerical solution can be solved using an explicit or an implicit method. Typically, the implicit method will work in all cases. The explicit method is recommended only when the implicit method fails to come to a solution.

Calculating Normal Vectors

The general equations for determining the normal vectors in the x, y, and z directions are:

$$\text{Normal vector in the x direction, } x = x / (x^2+y^2+z^2)^{.5}$$
$$\text{Normal vector in the y direction, } y = y / (x^2+y^2+z^2)^{.5}$$
$$\text{Normal vector in the z direction, } z = z / (x^2+y^2+z^2)^{.5}$$

The simplest way to calculate the unit vector is to draw an imaginary line at right angles (i.e., 90°) from the exposed surface of the target and to extend this imaginary line until it hits the walls, floor or ceiling of the compartment in which it is located. This imaginary line is actually a vector that starts at the surface of the exposed target and terminates on a wall, floor, or ceiling. Once the start point and the end point of a vector are known, the unit vector for this imaginary line may be calculated as follows:

$$x = (x_{end} - x_{start}) / ((x_{end} - x_{start})^2 + (y_{end} - y_{start})^2 + (z_{end} - z_{start})^2)^{0.5}$$
$$y = (y_{end} - y_{start}) / ((x_{end} - x_{start})^2 + (y_{end} - y_{start})^2 + (z_{end} - z_{start})^2)^{0.5}$$
$$z = (z_{end} - z_{start}) / ((x_{end} - x_{start})^2 + (y_{end} - y_{start})^2 + (z_{end} - z_{start})^2)^{0.5}$$

As an example, assume the following scenario:

- A square shaped target object is located in the middle of the floor of a compartment.
- The target is constructed from non-combustible materials except the top.
- The square shaped material is 1 m in depth, 1 m in breadth, and 1 m high.
- The reference point (0,0,0) in the compartment is the lower left hand side of the rear wall.
- The compartment is 3 meters in the x direction (i.e., the distance from the rear wall to the front wall of the compartment), 4 meters in the y direction (i.e., the distance from the left wall to the right wall of the compartment), and 5 meters in the z direction (i.e., the distance from the floor to the ceiling of the compartment).

Since the only side of the target that is combustible is the top surface of the target object, an imaginary line is draw perpendicular (i.e., at a 90° angle) from the top surface of the target object and extended until it reaches the outer boundary of the compartment. In this case, since the top of the target object is facing the ceiling, the imaginary line would run straight up to the ceiling, running parallel to the four walls of the compartment, and terminating at the ceiling at the point (1.5 m, 2 m, 5 m). Since the vector starts at point (1.5-m, 2-m, 1-m) and terminates at (1.5-m, 2-m, 5-m), the unit vectors can be calculated as follows:

$$x = (x_{end} - x_{start}) / ((x_{end} - x_{start})^2 + (y_{end} - y_{start})^2 + (z_{end} - z_{start})^2)^{.5}$$
$$= (1.5 - 1.5) / ((1.5 - 1.5)^2 + (2 - 2)^2 + (5 - 1)^2)^{.5} = 0$$

$$y = (y_{end} - y_{start}) / ((x_{end} - x_{start})^2 + (y_{end} - y_{start})^2 + (z_{end} - z_{start})^2)^{.5}$$
$$= (2 - 2) / ((1.5 - 1.5)^2 + (2 - 2)^2 + (5 - 1)^2)^{.5} = 0$$

$$z = (z_{end} - z_{start}) / ((x_{end} - x_{start})^2 + (y_{end} - y_{start})^2 + (z_{end} - z_{start})^2)^{.5}$$
$$= (5 - 1) / ((1.5 - 1.5)^2 + (2 - 2)^2 + (5 - 1)^2)^{.5} = 1$$

In this case, the unit vector is in the + Z direction.

Table 38. The TARG command (targets on compartment surfaces)

```
TARG Compartment Surface Depth Breadth Method Equation_Type

Example: TARG    1 DOWN   4.57000   7.62000
```

Parameter	Units	Default
Compartment: The compartment in which the target is located	none	none
Surface: Surface defines the compartment surface on which the target is located. It is one of the character strings, UP, FRONT, RIGHT, BACK, LEFT and DOWN	none	none
Position 1: Position of the target on the surface as a distance from the front wall (Z direction) of the target compartment for targets on the floor, ceiling, left, or right walls and from the left wall (X direction) for targets on the front or back wall (see figure).	m	½ depth
Position 2: Position of the target as a distance from the left wall (X direction) of the target compartment for targets on the floor or ceiling and from the floor (Z direction) for targets on the walls (see figure).	m	½ breadth
Method: Optional parameter that indicates the solution method. STEADY for steady state solution, XPLICIT for explicit solution, MPLICIT for implicit solution. The default is an implicit solution.	none	MPLICIT
Equation Type: If METHOD is not STEADY, this parameter further indicates the equation type to be either ODE or PDE. The default is PDE	none	PDE

Example:

```
TARGET   1   7.62000   1.52400   4.26720 0.000000   1.00000 0.000000
TARG    1 DOWN   4.57000   7.62000
```

4.7.2 Sprinklers and Detectors

Sprinklers and detectors are both considered detection devices by the CFAST model and are handled using the same input keywords. Detection is based upon heat transfer to the detector. Fire suppression by a user-specified water spray begins once the associated detection device is

activated. A maximum of 20 sprinklers or detectors can be included for any input file and model run. These can be in one compartment or scattered throughout the structure. The DETECT keyword is used for both detectors and sprinklers. The first three parameters are required. The next three have defaults. The last three must be specified if this is a heat detector or sprinkler.

Table 39. The DETECT command (heat detectors and sprinklers)

```
DETECT Type Compartment Activation_Temperature Depth Width Height RTI
Suppression Spray_Density

Example: DETECT  2  1    330.  2.5  4.55  4.59    50.  1 7.E-5
```

Parameter	Units	Default
Type: type of detector, use 1 for smoke detector and 2 for heat detector or sprinkler.	Integer	None
Compartment: the compartment in which the detector or sprinkler is located.	Integer	None
Activation temperature: the temperature at or above which the detector link activates.	K	None
Depth (Y): position of the detector or sprinkler as a distance from the front wall of the compartment (Y direction).	m	½ depth
Width (X): position of the object as a distance from the left wall of the compartment (X direction).	m	½ width
Height (Z): position of the object as a distance from the floor of the compartment (Z direction).	m	comp. height
RTI: the Response Time Index (RTI) for the sprinkler or detection device.	$(m \cdot s)^{1/2}$	none
Suppression: If set to a value of 1, the sprinkler will quench the fire with the specified spray density of water. If turned off with a value of 0, the device is handled as a heat or smoke detector only – values entered for activation temperature, RTI, and spray density are ignored.	Integer	none

```
DETECT Type Compartment Activation_Temperature Depth Width Height RTI
Suppression Spray_Density

Example: DETECT  2  1    330.  2.5  4.55  4.59    50.  1 7.E-5
```

Spray density: the amount of water dispersed by a water spray-sprinkler. The units for spray density are length/time. These units are derived by dividing the volumetric rate of water flow by the area protected by the water spray. The suppression calculation is based upon an experimental correlation by Evans [29], and depends upon the RTI, activation temperature, and spray density to determine the behavior of the sprinkler.	m/s	none

Example:

```
DETECT  2  1    330.370  2.50000  4.55000  4.59900    50.0000  1 7.00000E-005
```

This example defines a residential sprinkler in compartment 1 placed near the ceiling in the center of the compartment. The activation temperature of the sprinkler is 330 K, RTI of the sprinkler is 50 $m^{1/2}s^{1/2}$ and the spray density is $7x10^{-5}$ m/s.

- Often, the activation of smoke alarms is simulated with a temperature-based criterion. Davis and Notarianni [30] studied the activation of heat and smoke alarms in small and large compartments. They conclude that a temperature rise of approximately 5 ˚C corresponded to activation of ionization alarms for a range of fire sources and ceiling heights..
- Several cautions should be observed when using estimates of sprinkler suppression within the model: 1) the first sprinkler activated controls the effect of the sprinkler on the heat release rate of the fire. Subsequent sprinklers which may activate have no additional effect on the fire simulation. 2) The fire suppression algorithm assumes the effect of the sprinkler is solely to reduce the heat release rate of the fire. Any effects of the sprinkler spray on gas temperatures or mixing within the compartment are ignored. 3) The sprinkler always reduces the heat release rate of the fire. The ability of a fire to overwhelm an under-designed sprinkler is not modeled. 4) Since the dynamics of the sprinkler and direct effects of the spray on gas temperatures and velocities are not modeled, calculated times of activation of secondary sprinklers and / or detectors after the first sprinkler is activated should be not be modeled since the impact of the first sprinkler on the activation of additional sprinklers is not included in the CFAST model.

4.7.3 Defining Compartment Area

Two additional keywords, ROOMA and ROOMH, may be used for defining compartment properties for spaces which are not rectangular in area. The ROOMA and ROOMH keywords

57

allow the user to define non-rectangular rooms by specifying cross-sectional area as a function of height. One set of values is included for each compartment that has a variable cross-sectional area.

Table 40. The ROOMA and ROOMH commands (variable compartment cross-section)

```
ROOMA Compartment Number_of_Area_Values Area_Values

Example: ROOMA 1    3      10.0  5.0  3.0
```

Parameter	Units	Default Value
Compartment: The compartment in which the cross-sectional area varies as a function of height	none	none
Number_of_Area_Values: A varying cross-sectional area is specified by including the specified number of cross-sectional area values as a function of height, with matching height values specified with the ROOMH command. The number of area data points specified must equal the number of matching cross-sectional area values specified below. The number of values for the compartment must match those included for the same compartment on the ROOMH command, below.	none	none
Area Value(s): Values of cross-sectional area of the compartment as a function of height measured from the floor of the compartment. The values for the compartment correspond to height values included for the same compartment on the ROOMH command, below	m^2	none

```
ROOMH Compartment Number_of_Height_Values Height_Values

ROOMH 1    3      0.0   1.0  2.0
```

Parameter	Units	Default Value
Compartment: The compartment in which the cross-sectional area varies as a function of height	none	none

```
ROOMH Compartment Number_of_Height_Values Height_Values

ROOMH 1    3     0.0   1.0   2.0
```

Number of Height Data Points: A varying cross-sectional area is specified by including a specified number of height values with matching cross-sectional area values specified with the ROOMA command. The number of height data points specified must equal the number of matching cross-sectional area values specified above. The number of values for the compartment must match those included for the same compartment on the ROOMA command.	none	none
Height Data Point(s): Values of height for the corresponding cross-sectional area values measured from the floor of the compartment. The values for the compartment correspond to cross-sectional area values included for the same compartment on the ROOMA command.	m	none

Example:

```
ROOMA 1    3     10.0  5.0   3.0
ROOMH 1    3     0.0   1.0   2.0
```

The above example specifies that compartment 1 has a cross-sectional area of 10 m^2, 5 m^2 and 3 m^2 at elevations 0.0 m, 1.0 m and 2.0 m respectively.

4.7.4 Inter-compartment Heat Transfer (Conduction)

There are twor keywords that define physical connections which affect heat conduction between compartments. They are:

> CFCON   ceiling/floor heat conduction (vertical heat flow)
> HHEAT   heat transfer between connected compartment walls (horizontal heat flow)

Energy can be transferred from compartment to compartment through solid boundaries (walls, ceilings and floors) by means of conduction. The heat transfer between connected compartments is modeled by merging the connected surfaces for the ceiling and floor compartments or for the connected horizontal compartments. The heat conduction problem is solved for the merged walls using a temperature boundary condition for both the near and far wall. As before, temperatures are determined by the solver so that the heat flux striking the wall surface (both interior and exterior) is consistent with the temperature gradient at that surface. This option is implemented with the CFCON (for vertical heat transfer) and the HHEAT (for horizontal heat transfer) keywords.

Vertical Heat Transfer

Heat transfer between the ceiling and floor of specified compartments is included with the CFCON keyword. Ceiling to floor heat transfer occurs between interior compartments of the structure or between an interior compartment and the outdoors. The model checks to make sure that the ceiling and floor are reasonably contiguous (within 0.01 m). The assumption is made that this is true for the entire ceiling and floor.

Table 41. The CFCON command (vertical heat transfer between compartments)

`CFCON First_Compartment Second_Compartment`
 	
`Example: CFCON    1    2`

First Compartment: First of the connected compartments. Order of the inputs is not important
Second Compartment: Second of the connected compartments. Order of the inputs is not important

Example:

```
CFCON     1     2
```

- In order for this command to work, the CEILI and FLOOR (see section 4.4.2) properties must be specified so that conduction through ceilings and floors can be calculated.

Horizontal Heat Transfer

Horizontal conduction between specified compartments is included with the HHEAT keyword. Ceiling to floor heat transfer occurs between interior compartments of the structure or between an interior compartment and the outdoors.

There are three forms of this command. If the keyword is present, then the command takes one of three forms. The first is HHEAT with no arguments. In this case, CFAST computes heat conduction between every pair of rooms connected by a *horizontal flow* vent. The second form is to include a compartment number. In this case, CFAST will calculate the conductive heat transfer to all compartments connected to this compartment by horizontal convective flow. The third form specifies the compartments to be connected and what fraction of the compartment is connected to an adjacent compartment. This latter is particularly useful for rooms which are connected to adjacent rooms as well as hallways.

The user of the model is responsible for the consistency of these pairings. CFAST does not

check to insure that the specified compartment pairs are located next to one another.

Table 42. The HHEAT command (horizontal heat transfer between compartments)

HHEAT First_Compartment Number_of_Parts N pairs of {Second_Compartment, Fraction}		
Example: HHEAT 1 3 2 0.3 3 0.5 4 0.1		
Parameter	Units	Default
First Compartment: If First Compartment is specified, CFAST calculates heat conduction between that compartment and all compartments connected to First Compartment by horizontal convective flow. If omitted, heat conduction between all compartments connected by a horizontal flow vent.	none	All compartments
Number of Parts: Specify the fraction(s) of wall surface connecting the First Compartment and any number of other compartments. The number of parts specifies the number "second compartment" and "fraction" pairs to be specified below.	none	none
Second Compartment If the number of ordered pairs is specified, pairs of numbers which specify a connected compartment and the fraction of the vertical surface areas of the compartments which are connected can be specified. The second compartment specifies the compartment number of a compartment connected by a wall surface to the first compartment. One pair of numbers should be included for each pair of compartments to connect.	none	none
Fraction: If the number of ordered pairs is specified, pairs of numbers which specify a connected compartment and the fraction of the vertical surface areas of the compartments which are connected can be specified. The fraction specifies the fraction of the vertical surface area connecting the first and second compartment pair. One pair of numbers should be included for each pair of compartments to connect.	none	none

As an example, four cases were simulated by modeling all wall surfaces in a test case with gypsum and steel and by turning on and off the horizontal heat transfer option. The following table summarizes the results for these four cases. Each table entry gives the Upper room 1 temperature, upper room 2 temperature, upper wall temperature in room 1 and upper wall temperature in room 2.

	HHEAT Off	HHEAT On
Steel walls	1106/528.3/374.9/319	1118/535.3/399.6/340.2
Gypsum walls	1147/545.4/462.9/338	1148/545.8/464.1/339.3

The results are as one would expect. For each wall material turning HHEAT on leads to warmer calculated temperatures since the outside environment (where heat goes when HHEAT is off) is cooler than the adjoining room. The temperature differences are greater for steel than for gypsum since steel conducts heat more effectively.

4.7.5 The Ceiling Jet - An Additional Zone

Typically, zone models incorporate two zones in each compartment, one for the upper layer and one for the lower layer. A plume from a fire then pumps energy from the fire and lower layer into the upper layer. However, it has been observed that the heat loss from the upper layer to the ceiling and walls is higher than this paradigm would indicate. The cause is that a plume does not truly mix with the upper layer initially as indicated in the figure
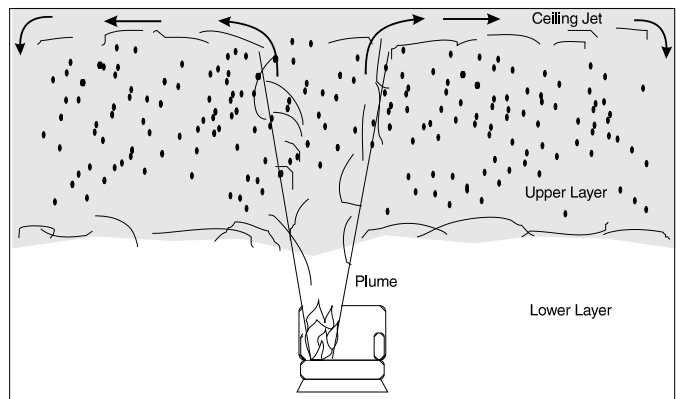
This leads to higher convective heat transfer than the mixed model would indicate. CFAST has incorporated a model of this additional heat loss based on the work of Cooper [31]. Heat loss by this mechanism to the ceiling is well established, the loss to walls is somewhat more uncertain due to the uncertainty of how far the plume descends below the ceiling.

To include ceiling jet effects in the calculation of the surface temperature of the ceiling, wall, or both surfaces, the CJET keyword is used. The ceiling jet calculation can be used for the ceiling, walls, or all surfaces. At present, this keyword effects only the calculation of the convective heating boundary condition for the conduction routines. If a particular surface is on, the ceiling jet algorithm is used to determine the convective heating of the surface. If CJET is off, the bulk temperature of the upper layer determines the convective heating.

Table 43. The CJET command (ceiling jet)

```
CJET Ceiling_Jet_Flag
Input:      OFF, CEILING, WALL, or ALL

Example: CJET CEILING
```

Parameter	Units	Default
Ceiling Jet Flag: directs the model to calculate the effects of a ceiling jet in all compartments containing a fire. The value of the flag may be OFF (to not calculate ceiling jets), CEILING (to include calculations for ceiling surfaces only, WALL (to include calculations for wall surfaces only), or ALL (to include calculations for both ceiling and wall surfaces).	none	OFF

Example:

```
CJET CEILING
```

This command directs the program to use the ceiling jet algorithm for all compartment ceiling heat transfer calculations.

• Using this option affects all fires and it is suggested that this option be turned on and set to "CEILING."

4.7.6 Utility Commands

Occasionally, it is necessary to start a calculation with default values different from those typically used by the model. There are several utility commands to control execution of the model:

INTER	to set the intial layer interface position in all compartments
DTCHE	prevent excessive computation
SETP	set initial values for temperature, pressure, and interface height
STPMAX	set initial and maximum time increments

The initial interface height is normally at the ceiling of a compartment. When it is desired to set it at some other value, use INTER. Each invocation can have as many entries as desired, but they must be in pairs of (compartment $\leq N_R$, height).

Table 44. The INTER command (set initial interface height)

<table>
<tr><td colspan="3">

```
INTER Initial_Interface_Height_1 Initial_Interface_Height_2 ...
Initial_Interface_Height_N

Example:  INTER 2    1.2      3    2.0
```
</td></tr>
<tr><td>Parameter</td><td>Units</td><td>Default</td></tr>
<tr><td>Initial_Interface_Height_1 Initial_Interface_Height_2 ...
Initial_Interface_Height_N: specifies the initial height of the layer interface in each compartment of the simulation. The number of values on the line must equal the number of compartments in the simulation. By default, CFAST sets the layer thickness to a small value to avoid numerical instabilities calculating layer volume as the thickness approaches 0. Normally, this command would not be used.</td><td>m</td><td>0.999 ceiling height</td></tr>
</table>

Example:

```
INTER 2   1.2     3   2.0
```

The purpose of DTCHE is to prevent excessive computation with a very small time step. This often appears to users as a stalling condition, when it is simply the set of equations has reached a point that requires a very small increment in time for the solver to converge. Note that if DASSL cannot converge it will quit by itself. However, there is no lower limit.

Table 45. The DTCHE command (set minimum time step and maximum iteration count)

<table>
<tr><td colspan="3">

```
DTCHE Minimum_Time_Step Maximum_Iteration_Count

Example: 1.0e-10    1000
```
</td></tr>
<tr><td>Parameter</td><td>Units</td><td>Default</td></tr>
<tr><td>Minimum time step: smallest value the model will use to advance the solution of the equation set</td><td>s</td><td>1.0×10^{-9}</td></tr>
<tr><td>Maximum iteration count: Maximum number of iterations the model will use at this minimum time step in attempting to advance the solution of the equation set.</td><td>Count</td><td>100</td></tr>
</table>

Example:

```
DTCHE 1.0e-10 1000
```

would stop CFAST after 1000 iterations of the solver below a time step of 1.0×10^{-10}.

The purpose of SETP is to allow variables to be set so that the model can be updated in real time.

Table 46. The SETP command (set initial parameters)

SETP File_Name Example: SETP newdata.p0		
Parameter		default
File Name: File name for setting internal values. This keyword should be used with caution since it does not necessarily set the model consistently. The file named in the SETP command has a special format. The first line must be the name of the input data file referencing the SETP set, followed by lines specifying the conditions		none
Compartment Value {'L' or 'U'} {'PRESS' TEMP' or 'INTER'}		

For example, if one were using the input data file `DATA.DAT` and it had the entry:

```
SETP newdata.p0
```

then newdata.p0 might look something like

```
FILE DATA.DAT
1 230. U TEMP
2 101303. L PRESS
```

The purpose of STPMAX is to limit the maximum time step that will be taken by the solver. Generally, the numerical solver in CFAST will take as big a time step as it can so long as there is convergence. In many cases this works well. However, the descritization in CFAST is on one second intervals. If fire events change (for example, a door is opened), the prediction can go beyond the desired time and the code has to backtrack to the beginning of the event. This can slow the computation. In order to prevent the model from slowing down in many cases, the maximum time step is set to one second. Larger values are okay, so long as the model converges.

Table 47. The STPMAX command (set maximum time step)

STPMAX Maximum_Time_Step		
Example: STPMAX 5.0		
Parameter	Units	Default
Maximum time step: Largest time interval that the model will use in attempting to advance the solution of the equation set. The model may use smaller values if a solution cannot be obtained at this value.	s	1

Example:

```
STPMAX 5.0
```

4.8 Scenario and Software Limits

CFAST is intended for use with a wide variety of fire scenarios. A number of limits to the inputs in the software implementation of the model are noted below. Details of all of the inputs for the program are included in Chapter 4.

Table 48. Scenario and software limits for the CFAST model

Maximum simulation time in seconds	86 400
Maximum number of compartments	30
Maximum number of fires which can be included in a single test case (including the main fire)	31
Maximum number of fire definitions which can be included in a single fire database file	30
Maximum number of data points for a single main or secondary object fire	21
Maximum number of material thermal property definitions which can be included in a single thermal database file	56
Maximum number of slabs in a single surface material in the thermal database file	3

Maximum total number of fans in all mechanical ventilation systems which can be included in a single test case	5
Maximum total number of ducts in all mechanical ventilation systems which can be included in a single test case	62
Maximum total number of connections between compartments and mechanical ventilation systems which can be included in a single test case	62
Maximum number of independent mechanical ventilation systems which can be included in a single test case	10
Maximum number of targets which can be included in a single test case. In addition, the CFAST model includes a target on the floor of each compartment in the simulation and one for each object fire in simulation.	30
Maximum number of data points in a history or spreadsheet file	900

5 Output

5.1 Overview

The output of CFAST are the sensible variables that are needed for assessing the environment in a building subjected to a fire. These include temperatures of the upper and lower gas layers within each compartment, the ceiling/wall/floor temperatures within each compartment, the visible smoke and gas species concentrations within each layer, target temperatures and sprinkler activation time. The amount of information can be very large, especially for complex geometries and long simulations.

The default output is called the compact form, and shows the basic information about a scenario, including layer temperatures and the size of fires. This can be duplicated from other programs with the /r:c option. Default text output provides a simple overview for the user to make sure the case runs as expected. Using the /r reporting options allow the user to obtain additional details of the simulation results.

The compact output for each compartment is listed across the screen and the compartments are listed down the screen starting with compartment one and ending with the outside. The example below shows the initial and final conditions for a calculation:

```
Time =      0.0 seconds.

Compartment   Upper    Lower    Inter.   Pyrol    Fire
              Temp.    Temp.    Height   Rate     Size
              (K)      (K)      (m)      (kg/s)   (W)
-----------------------------------------------------------
    1         300.0    300.0    2.3      1.400E-03 2.530E+04
    2         300.0    300.0    2.3      0.000     0.000
 Outside                                           0.000


Time =    200.0 seconds.

Compartment   Upper    Lower    Inter.   Pyrol    Fire
              Temp.    Temp.    Height   Rate     Size
              (K)      (K)      (m)      (kg/s)   (W)
-----------------------------------------------------------
    1         1252.    475.1    0.63     0.278     4.831+06
    2         828.7    325.4    1.0      0.000     0.000
 Outside                                           1.609E+03
```

The first column contains the compartment number. On each row with its compartment number from left to right is the upper layer temperature, lower layer temperature, the height of the

interface between the two layers, the total pyrolysis rate, and finally the total fire size. The only value given for the outside is the total heat release rate of fires venting to the outside.

5.1.1 Detailed Outputs

The reporting options are `/R:WINFST`

The options are listed consecutively without commas or spaces between. The option is not case sensitive and order in the option list is not significant. When invoked the reporting option overrides the default output and reports only the information that the user requests. The following sections describe each of the options. Each section refers to a specific part of the print out at the end of the chapter and appears in the order the output appears. A description of each option follows.

5.1.2 Output for Option /R:I

To get a summary of the input file, use /R:I. This option prints the initial conditions to the output before the actual run starts. This merely mimics the inputs specified by the user in the input data file The initial conditions brake down into seven sections. Each is described below with the section name. The following explanation uses the output from the case `STANDARD.DAT` which is captured using the program report. `STANDARD.DAT` is included in the distribution. Please note, there are not mechanical ventilation or horizontal vents in `STANDARD.DAT`, so below the section saying there are no mechanical ventilation connections, there is an annotated section from additional data files, `TESTVVENT.DAT,  TESTHVAC.DAT` and `TESTSPEC.DAT`.

Title: The title line consists of the version of CFAST and the title included in the file. The first line of the output is the title line. It is the only section of the initialization output without a section header. The Checksum at present doesn't do anything. It is for future verification that a file has not been changed.

```
**      CFAST  Version  5.1.1  Run 10/25/2004    **
**                                               **
**            A contribution of the              **
** National Institute of Standards and Technology **
**           Gaithersburg, MD  20899             **
**           Not subject to Copyright            **

Data file is standard.dat    (Checksum 00000000)
```

Overview: The overview gives a general description of the case. The output is fairly self explanatory. Doors, ..." is the total number of horizontal natural flow vent connections in all compartments of the simulation. "Ceil. Vents, ..." gives the total number of vertical natural flow vent connections in all compartments of the simulation. The last header on the line "MV Connections" has the total number mechanical flow connections to all compartments in the simulation. Times in these outputs come from the TIMES input. All times are in s.

```
OVERVIEW

Compartments    Doors, ...    Ceil. Vents, ...    MV Connects

   1               1              0                   0

Simulation     Print      History     Restart
   Time        Interval   Interval    Interval
   (s)           (s)        (s)         (s)

  1800           10         10           0

Ceiling jet is on for all
History file is STANDARD.HI
```

Ambient Conditions: This section, like the overview section, needs little elaboration. It gives the starting atmospheric conditions for the simulation both for outside and inside the structure. Data for these outputs come from the TAMB and EAMB inputs. Temperatures are in K, pressure in Pa, elevations in m, and wind speed in m/s. Wind Power is the dimensionless power law coefficient from the WIND input.

```
AMBIENT CONDITIONS

Interior       Interior    Exterior       Exterior     Station      Wind      Wind         Wind
Temperature    Pressure    Temperature    Pressure     Elevation    Speed     Ref. Height  Power
(K)            (Pa)        (K)            (Pa)         (m)          (m/s)     (m)

   300.        101300.        300.        101300.        0.00        0.0        10.0        0.16
```

Compartments: The compartments section gives a summary of the geometry for the simulation. A simple table summarizes the geometry with compartments running down the page in numerical order. The various dimensions for each compartment are on the row with its compartment number. Two columns need explanation. The second to last column "Ceiling Height" gives the height of the ceiling relative to the station height in the Ambient Conditions section. Similarly the "Floor Height" refers to the height of the floor above the station height.

```
COMPARTMENTS

Compartment   Width    Depth    Height    Area     Volume    Ceiling   Floor
                                                             Height    Height
              (m)      (m)      (m)       (m^2)    (m^3)     (m)       (m)
------------------------------------------------------------------------------
    1         9.10     5.00     4.60      45.50    209.30    4.60      0.00
```

- Values for width, depth, height, and floor height come directly from the WIDTH, DEPTH, HEIGH, and HI/F specifications in the input data file.

- Area is calculated as width x depth in m^2.

- Volume is calculated as width x depth x height in m^3.

- Floor height is the absolute elevation of the ceiling of the compartment and is just station elevation (from the ambient conditions section) + compartment height in m.

Vent Connections: Vent connections further divides into three distinct parts. Each part has its own sub heading. A description of each follows.

Horizontal Natural Ventilation is the first table in the vent connections sections. Each row in the table characterizes one vent. The first two columns contain the two compartments connected by the vent. Each vent is ordered first by the lower number of the two compartments and then the numeric order of the second compartment. The third column gives the vent number. Column four is the width of the vent. The next two columns report the sill and soffit height for the vent relative to the floor of the first compartment. The seventh and eighth columns have a second listing of the sill and soffit height, this time relative to the station height. Area of the vent is in the last column.

```
VENT CONNECTIONS

Horizontal Natural Flow Connections (Doors, Windows, ...)

From          To          Vent    Width    Sill     Soffit   Abs.     Abs.     Area
Compartment   Compartment Number           Height   Height   Sill     Soffit
                          (m)     (m)      (m)      (m)      (m)      (m)      (m^2)
-------------------------------------------------------------------------------------
    1         Outside      1      2.40     0.00     1.00     0.00     1.00     2.40
```

- From compartment, to compartment, vent number, width, sill height, and soffit height all come directly from the HVENT specifications in the input data file.

- Absolute sill height and absolute soffit height is the station elevation + compartment floor height + sill height. Absolute soffit height and absolute soffit height is the station elevation + compartment floor height + soffit height

Vertical Natural Ventilation prints out in a similarly simple table. Again each vent is one row of the table. The first column is the upper compartment. The upper compartment is the compartment where the vent opens into the floor. The second column is the lower compartment where the vent is in the ceiling. The third column describes the shape of the vent, which can be either round or square. The fourth column gives the area of the vent. The last two columns are the height of the vent, relative to the floor of the lower room and relative to the station height respectively.

This is the output from STANDARD.DAT

```
There are no vertical natural flow connections

There are no mechanical flow connections
```

This is an alternate example that includes a vertical flow connection from TESTVVENT.DAT

```
Vertical Natural Flow Connections (Ceiling, ...)


Top           Bottom        Shape    Area      Relative  Absolute
Compartment   Compartment                      Height    Height
                                      (m^2)     (m)       (m)
------------------------------------------------------------------
   2             1           Round    0.10      2.30      2.30
```

which shows a circular hole in the ceiling of area 0.1 m^2.

- Top compartment, bottom compartment, shape, and area come from the VVENT specifications in the input data file.

- Relative height is the height of the vent above the floor of the bottom compartment and absolute height is the height of the vent above the station elevation.

Mechanical Flow Connections uses two tables. The first table lists all the connections to compartments and ducts. Each row is either a connection or duct. The first column tells to which system the duct or connection belongs. A system is a set of continuously linked connections, ducts and fans. The example contains only one system. If a second system had been part of the file, a line would have been skipped before starting into the second system. The second and fourth columns give both ends of either the connection or duct. The words "comp" or "node" proceeds the numbers in these two columns. "Comp" means a compartment number. Rows with a compartment number in either the "from" or "to" columns are connections to compartments, which are basically vents. The rest of the rows define ducts. The third and fifth columns give the elevations of the proceeding columns. The sixth column is the length which only applies to ducts. Connections do not have a length so the column is left blank. Next comes the cross sectional area. The last column tells the absolute roughness. This value is the resistance the duct provides to the flow of air. Like the length, this value only has meaning for a duct so it is blank for a connection. Air flows both ways though connections and ducts. The headers "from" and "to" say nothing about the direction of air flow. The headers only provide a convenient way of listing both ends of a duct or connection.

```
Mechanical Flow Connections (Fans, Ducts, ...)

Connections and Ducts


System   From          From     To         To        Length   Area      Rough
                       Elev.                Elev.
                       (m)                  (m)       (m)      (m^2)     (mm)
--------------------------------------------------------------------------------
   1     Comp  1       2.10     Node  1     2.10               0.12
         Node  1       2.10     Node  2     0.00      2.30     0.08      0.00
```

- All of the inputs come from the MVOPN and MVDUCT specifications in the input data file.

The second table lists the fans. The first five columns in the fan table appear almost the same as the connections and ducts table. The table lists, in order, the number of the system the fan is a

73

part, the "from" node and its height, the "to" node and its height. A fan actually draws air from the first or "from" node and pushes it to the second or "to" node. In the second table, the headers give the direction of the flow of air. The sixth column is the fan number as defined in CEdit. The next two columns are the minimum and maximum pressures at which the fan curve is defined. The rest of the row is made up of the two to five fan curve coefficients in the input file.

```
Fans


System    From        From      To         To      Fan      Minimum   Maximum   Fan Curve
                      Elev.                Elev.   Number
                      (m)                  (m)              (Pa)      (Pa)
-------------------------------------------------------------------------------------
  1      Node  2      0.00    Node  3      2.10      1        0.00    300.00    0.14     -4.33E-04
         Node  3      2.10    Outside      2.10               0.12
```

- All of the inputs come from the MVFAN specifications in the input data file.

Thermal Properties The thermal properties section brakes into two parts. The first part is a table that lists the material for each surface of each compartment. The compartments appear as rows down the page in numerical order. From left to right next to the compartment number comes the material for the ceiling, wall and floor. The second part lists the entries in the thermal data base. The first line gives the database file used. Next comes a listing of each material used. In addition to materials for compartment surfaces, any materials specified for targets are also listed. For each listing of a material, the name is followed by the conductivity, specific heat, density, thickness and emissivity. Additionally, the HCl constants used to calculate the HCl deposition for the surface are displayed.

```
THERMAL PROPERTIES


Compartment    Ceiling     Wall       Floor
----------------------------------------------------------------------
    1          GLASSFB3    CONCRETE    CONCRETE

Thermal data base used: THERMAL.DF

Name     Conductivity Specific heat    Density    Thickness   Emissivity             HCL B's (1->5)
GLASSFB3    3.600E-02     795.          105.       1.300E-02    0.900     0.00E+00  0.00E+00  0.00E+00  0.00E+00  0.00E+00
CONCRETE     1.75       1.000E+03     2.200E+03    0.150        0.940     0.00E+00  0.00E+00  0.00E+00  0.00E+00  0.00E+00
PLYWOOD      0.120      1.215E+03      545.        1.300E-02    0.900     0.00E+00  0.00E+00  0.00E+00  0.00E+00  0.00E+00
```

- Material choices of the ceiling, walls, and floors come from the CEILI, WALLS, and FLOOR specifications in the input data file.

- Materials properties for the chosen materials come from the specified thermal properties file.

- Units for thermal properties are standard S.I. units. For thermal conductivity, W/m K; for specific heat, J/kg K; for density, kg/m^3; for thickness, m; emissivity is dimensionless.

Targets: The entry for targets shows the orientation of additional targets specified in the data file. Note that the first two are always present. The third one is the target specified in the data file. Each target is numbered based on the order of the target specifications in the input data file. The compartment number, position of the target within the compartment, direction of the front face of the target object expressed as a normal unit vector to the surface, and object material.

```
TARGETS


Target Compartment     Position (x, y, z)        Direction (x, y, z)       Material
-----------------------------------------------------------------------------------
   1       1          2.20    1.88    2.34      0.00    0.00    1.00    CONCRETE
   2       1          2.15    1.65    0.00     -1.00    0.00    0.00    PLYWOOD
   3       1          2.50    4.55    0.00      0.00    0.00    1.00    CONCRETE   Floor, compartment  1
```

- All of the inputs for targets come from the TARGET or TARG commands in the input data file. Direction is specified as a unit vector as described in the section on target input.

- Units for position and direction are all in m.

Fires The fire section lists all the information about the main fire and any object fires that might exist. All the information for each fire is listed separately. If there is a main fire, it comes first. Each fire listing has the same form. First is the name of the fire followed by a list of general information. Listed left to right is the compartment the fire is in, the type of fire, the x, y, z position, the relative humidity, the lower oxygen limit, and finally the pyrolysis temperature. A table of time history curves for the fire follows. The table contains all the time history curves for the fire. Each row on the table is a specific time given in the left most column. The rest of the columns give the values at that particular time. The column headers are keywords used in the datafile. The keyword meanings are: fmass is pyrolysis rate; hcomb is heat of combustion; fqdot is heat release rate; fhigh is height of fire; C/CO2 is carbon to carbon dioxide ratio; CO/CO2 is the carbon monoxide to carbon dioxide ratio; H/C is the hydrogen to carbon ratio; O/C is the oxygen to carbon ratio; HCN is the hydrogen cyanide production rate; and HCl is the hydrogen chloride production rate.

```
FIRES

Name: Main Fire

Compartment     Fire Type      Position (x,y,z)     Relative    Lower O2    Pyrolysis
                                                    Humidity    Limit       Temperature
    1           Constrained    3.00   5.70   0.00   50.0        12.00          300.

  Time     Fmass     Hcomb     Fqdot     Fhigh    C/CO2     CO/CO2    H/C       O/C       HCN       HCL
  (s)      (kg/s)    (J/kg)    (W)       (m)      (kg/kg)   (kg/kg)   (kg/kg)   (kg/kg)   (kg/kg)   (kg/kg)
------------------------------------------------------------------------------------------------------------
    0.      0.0      2.41E+07   0.0      0.0       0.0       0.0      0.16      0.0       0.0       0.0
   60.      8.30E-03 2.41E+07  2.00E+05  0.0       0.0       0.0      0.16      0.0       0.0       0.0
  120.      1.45E-02 2.41E+07  3.50E+05  0.0       0.0       0.0      0.16      0.0       0.0       0.0
  180.      1.41E-02 2.41E+07  3.40E+05  0.0       0.0       0.0      0.16      0.0       0.0       0.0
  240.      8.30E-03 2.41E+07  2.00E+05  0.0       0.0       0.0      0.16      0.0       0.0       0.0
  300.      6.22E-03 2.41E+07  1.50E+05  0.0       0.0       0.0      0.16      0.0       0.0       0.0
  360.      4.15E-03 2.41E+07  1.00E+05  0.0       0.0       0.0      0.16      0.0       0.0       0.0
```

Time (s)	Fmass (kg/s)	Hcomb (J/kg)	Fqdot (W)	Fhigh (m)	C/CO2 (kg/kg)	CO/CO2 (kg/kg)	H/C (kg/kg)	O/C (kg/kg)	HCN (kg/kg)	HCL (kg/kg)
420.	4.15E-03	2.41E+07	1.00E+05	0.0	0.0	0.0	0.16	0.0	0.0	0.0
480.	3.32E-03	2.41E+07	8.00E+04	0.0	0.0	0.0	0.16	0.0	0.0	0.0
540.	3.11E-03	2.41E+07	7.50E+04	0.0	0.0	0.0	0.16	0.0	0.0	0.0
1800.	4.15E-03	2.41E+07	1.00E+05	0.0	0.0	0.0	0.16	0.0	0.0	0.0

Name: WARDROBE   Referenced as object # 1

Compartment	Fire Type	Position (x,y,z)			Relative Humidity	Lower O2 Limit	Pyrolysis Temperature
1	Constrained	2.15	1.65	0.00	50.0	12.00	300.

Time (s)	Fmass (kg/s)	Hcomb (J/kg)	Fqdot (W)	Fhigh (m)	C/CO2 (kg/kg)	CO/CO2 (kg/kg)	H/C (kg/kg)	O/C (kg/kg)	HCN (kg/kg)	HCL (kg/kg)
0.	0.0	1.59E+06	0.0	0.0	0.0	0.0	0.33	0.0	0.0	0.0
70.	3.14E-02	1.59E+06	5.00E+04	0.0	1.60E-02	0.23	0.33	0.0	0.0	0.0
100.	8.17E-02	1.59E+06	1.30E+05	0.0	1.90E-02	0.13	0.33	0.0	0.0	0.0
140.	0.33	1.59E+06	5.30E+05	0.0	4.00E-03	9.00E-03	0.33	0.0	0.0	0.0
190.	1.57E-02	1.59E+06	2.50E+04	0.0	0.0	1.60E-02	0.33	0.0	0.0	0.0
200.	3.77E-02	1.59E+06	6.00E+04	0.0	0.0	3.10E-02	0.33	0.0	0.0	0.0
340.	1.57E-02	1.59E+06	2.50E+04	0.0	0.0	6.30E-02	0.33	0.0	0.0	0.0
1170.	3.10E-03	1.61E+06	5.00E+03	0.0	0.0	0.0	0.33	0.0	0.0	0.0

- All of the inputs for the main fire come from the fire specifications in the input data file. Data for the object fire comes from the object data file included with the CFAST software.

- Units for most values are included in the output. Fire position is in m, relative humidity is in %, lower oxygen limit is in volume percent, and pyrolysis temperature is in K.

5.1.3 Output for Option /R:N

An expanded version of the compact default output called the normal print out can be obtained using the /R:N option. When requested, the normal print out is the first information printed at each interval. The first part of the print out looks similar to the default print out. It is laid out along the same lines, with the calculated values for each compartment listed left to right and compartments listed down. The left most column has the compartment. The next three columns also give the same values as the default output. They are upper layer temperature, lower layer temperature and interface height. The output changes at this point. The fifth column gives the upper layer volume. In the sixth, in parenthesis, is the percent of the total compartment volume the upper layer takes up. The pressure difference from ambient follows in the seventh column. The rest of the columns are the temperature of the surfaces that are turned on in K.

Time =   1800.0 seconds.

Compartment (K)	Upper Temp.	Lower Temp. (K)	Inter. Height (m)	Upper Vol. (m^3)	Upper Absorb (m^-1)	Lower Absorb (m^-1)	Pressure (Pa)	Ambient Target (W/m^2)	Floor Target (W/m^2)
1	377.0	303.8	0.7236	1.76E+02(84%)	1.000E-02	1.000E-02	-9.483E-02	437.	335.

The second table of the normal print out has information about the fires. In essence it is two tables joined. The first part lists information by fire. It starts with the main fire, if there is one, and then the object fires down the page. The fires are listed in the second column followed by the plume flow rate, the pyrolysis rate and the fire size. The next three columns are then skipped. The next column with information is the amount of heat given off by each fire convectively, followed by the amount of heat given off radiantly. The second part starts after all the fires have been individually listed. It gives the totals for all fires in each compartment. The first column has the compartment number. The compartments start at one and are listed down the page in order. The third to fifth columns are the same as the first part except the values are totals for the compartment and not just for one fire. The sixth column has the total heat release rate that occurs in the upper layer. The next column has the same total in the lower layer. The eighth column has the total size of vent fires in the compartment. The last two columns of the table gives the convective and radiative parts of the fire heat release.

```
Fires


Compartment    Fire     Plume    Pyrol    Fire     Flame    Fire in  Fire in  Vent     Convec.   Radiat.
                        Flow     Rate     Size     Height   Upper    Lower    Fire
                        (kg/s)   (kg/s)   (W)      (m)      (W)      (W)      (W)      (W)       (W)
--------------------------------------------------------------------------------------------------------
               Main     0.291    4.149E-03 9.998E+04 1.11                                       6.999E+04 3.000E+04
            WARDROBE    5.026E-02 3.100E-03 4.988E+03 0.120                                     3.492E+03 1.496E+03

     1                  0.341    7.249E-03 1.050E+05           0.00     1.050E+05 0.00
```

- Flame height is calculated from the work of Heskestad [32]. Valid for a wide range of hydrocarbon and gaseous fuels, the correlation is given by

$$H = -1.02 + 0.235 Q_f^{2/5}$$

   where H is the average flame height and Q_f is the fire size. The mean flame height is defined as the distance from the fuel source to the top of the visible flame where the intermittency is 0.5. A flame intermittency of 0.5 means that the visible flame is above the mean 50 % of the time and below the mean 50 % of the time.

5.1.4 Output for Option /R:W

The /R:W option provides two tables displaying information about wall surface or target temperatures and fluxes, and heat detectors or sprinklers. The left most column specifies the compartment number; followed by four columns providing the temperatures of the bounding surfaces of the compartment in contact with the ceiling, upper wall surface (in contact with the upper layer gases), lower wall surface (in contact with the lower layer gases), and floor, in that order. Next comes information about targets in the compartment, with each target listed on a separate line. Information in the columns includes the surface temperature of the target, net heat flux to the target, and the percentage of that net flux that is due to radiation from the fire, radiation from compartment surfaces, radiation from the gas layers, and convection from the gas

77

surrounding the target. CFAST includes one target in the center of the floor for all compartments. Information on additional targets specified by the user in the input data file are also included, in the order specified in the input file.

```
Surfaces and Targets

Compartment   Ceiling   Up wall   Low wall   Floor    Target    Target   Flux To    Fire     Surface   Gas
              Temp.     Temp.     Temp.      Temp.              Temp.    Target     Rad.     Rad.      Rad.     Convect.
              (K)       (K)       (K)        (K)                (K)      (W/m^2)    (%)      (%)       (%)      (%)
-------------------------------------------------------------------------------------------------------------------
     1        339.2     312.3     306.3      310.8    Floor     311.    863.4      0.0      27.9      74.7     -2.6
                                                       1        320.    1318.      0.0       7.7      71.7     20.6
                                                       2        367.    647.8      0.0      25.9     120.5    -46.4
```

The second table provides information about heat detectors and sprinklers included in the input data file. For each output time, the output includes the sensor temperature, whether the sensor temperature has exceeded its specified activation temperature, and the temperature and velocity of the gas adjacent to the sensor. For sensors placed near the ceiling of a compartment with a fire, the temperature and velocity are those of the ceiling jet; for other compartments, it is the upper layer temperature and a default velocity of 0.1 m/s.

```
Sensors

                             Sensor                      Smoke
Number  Compartment   Type   Temp (K)   Activated    Temp (K)   Vel (M/S)
------------------------------------------------------------------------
  1         1         HEAT    3.149E+02  YES          3.136E+02  4.152E-01
```

- Temperatures are all in K, fluxes in w/m^2, and velocities in m/s.

- In all cases, the flux to/from a target is net radiation or net convection. That is, it is the incoming minus the outgoing. So while a target or object is heating, the flux will be positive, and once it starts to cool, the flux will be negative.

- Values for radiation from fires (fire rad.), radiation from surfaces (surface rad.), radiation from the gas layers (gas rad.), and convection from surfaces (convect) are expressed as a percentage of the net flux to target (flux to target). Positive values indicate heat gains by the target and negative values indicate heat losses.

5.1.5 Output for Option /R:S

The /R:S option has two tables displaying information about the amounts of species in each layer. The species information follows the normal print out. The first table gives species concentrations for the upper layers of all the compartments and the second reports the same for the lower layers of all the compartments. Again the compartments are listed down the page and the information across the page. The species are each given in one of several different terms.

78

Below each header is the units for the given value. Most of the headers are simply the chemical formula for the species being tracked. However, a couple are not obvious. "TUHC" is the total unburned hydrocarbons or the pyrolyzed fuel that hasn't burned yet. "OD" is the obscuration density, which is a measure of the amount of smoke. The last four columns, "HCl c", "HCl f", "HCl uw" and "HCl lw", are the amount of HCl deposition on the ceiling, floor, upper wall and lower wall respectively.

```
Time =   1800.0 seconds.

Upper Layer Species

Compartment   N2      O2      CO2     CO      HCN     HCL     TUHC    H2O     OD      CT
              (%)     (%)     (ppm)   (ppm)   (ppm)   (%)     (%)     (%)     (1/m)   (g-min/m3)
------------------------------------------------------------------------------------------------
    1         77.5    18.1    1.24    777.    568.    415.    0.00    2.55    7.26    353.

Lower Layer Species

Compartment   N2      O2      CO2     CO      HCN     HCL     TUHC    H2O     OD      CT

              (%)     (%)     (%)     (ppm)   (ppm)   (ppm)   (%)     (%)     (1/m)   (g-min/m3)
------------------------------------------------------------------------------------------------
    1         79.3    20.7    7.209E-08 4.531E-05 3.312E-05 2.417E-05 0.00   7.677E-07 4.830E-07 0.860
```

- The report by species for nitrogen, oxygen, hydrogen chloride and the total unburned hydrocarbons (fuel vapor in the layer) are percent by volume. Carbon dioxide, carbon monoxide and hydrogen cyanide are in parts per million, which is also a volume fraction. Optical depth per meter is a measure of the visibility in the smoke. This is covered in detail in the comment on visibility in the section on fires and species specification. The concentration-time (CT) calculation is an integration of the species input for type CT (See section 4.6.1 for the input of CT) and is intended to represent a relative dose of toxic gas species [33].

5.1.6 Output for Option /R:F

Information about hallway and vent flow is obtained by using this option. It includes two sections, one with information about hallway flow specified with the HALL command, and a larger section detailing mass flow through horizontal, vertical, and mechanical vents. For hallway flow, the output begins with the compartment specified as a hallway in the input data file, followed by the time at which gas began to flow into the hallway through the vent that connects the hallway to the main fire compartment. Following the start time are the average gas velocity, the layer depth of the initial flow down the corridor, and the distance from the end of the corridor that the hot layer is estimated to have reached.

```
Hall Flow

Compartment   Start Time    Velocity      Depth       Distance
              (s)           (m/s)         (m)         (m)
------------------------------------------------------------------
    3         2.59          1.69          0.573       32.3
```

The section for vent flow is titled "Flow Through Vents (kg/s)." Because flow is always given in positive values, each vent is listed twice. Once for flow going from compartment A to compartment B and a second time for flow from B to A. As the example below shows, the first column lists the compartment. The second column specifies the vent, including the type of vent (an "H" in this column stands for horizontal flow, such as through a doorway or window; a "V" here would mean vertical flow, such as through an opening in the ceiling, and an "M" stands for a mechanical ventilation connection) and the compartment from which the flow comes. Up to six additional columns detail the flow at this vent. Flow into and out of the compartment through the vent in the upper and lower layers are included, along with mixing between layers at the vent into the upper layer and into the lower layer.

```
Flow Through Vents (kg/s)

To            Through          Upper Layer            Lower Layer          Mixing       Mixing
Compartment   Vent          Inflow     Outflow     Inflow     Outflow     To Upper     To Lower
---------------------------------------------------------------------------------------------------
   1          H Outside #1              0.344       0.401      6.416E-02

Outside       H Comp  1 #1   0.344                  6.416E-02  0.401
```

- The mass balance is the sum of the flow in minus the flow out. (Note that this is an extended run to achieve results close to steady state.) For any compartment, this is just ("Upper Layer Inflow" + "Lower Layer Inflow" + "Pyrol Rate") – ((Upper Layer Outflow + Lower Layer Outflow) with the inflow and outflow summed for each vent. Note that the mixing flows are ignored since they are just exchanged between layers in the same compartment. For the above example, the mass balance is:

$(0.0 + 0.401 + 7.2 \times 10^{-3})$ kg/s – $(0.344 + 6.4 \times 10^{-2})$ kg/s = -0.002 kg/s

where the pyrolysis rate (from the"normal" output) is 7.2×10^{-3} kg/s. The result is about the right magnitude (about 0.5% of the mass flow into or out of the vent) for net mass loss. The mass loss should still be slightly negative since the compartment continues to heat.

5.2 Spreadsheet Output

CFAST and REPORTSS generate the same type of output as report. However, the format is a comma delimited file so that it can be easily imported into a spread sheet. The caution is that reportss writes all available data to the spreadsheet which can result in more columns than spread sheets can handle. So it is recommended that only data of interest be written, rather than all options. The full set is /r:winfs. The recommendation would be to use /r:n, then /r:f and so on. This keeps the spread sheet to a manageable size.

The headings in the comma delimited file generally spell out the information that is in the

column. An example from `STANDARD.DAT` is

```
row 1: "TIME","Upper Layer Temp(  1)","Lower Layer Temp(  1)","Layer Height(  1)","Upper Layer Volume(  1)","Pressure(
1)","Ambient Target(  1)","Floor Target(  1)","Main Plume Flow","Main Pyrolysis Rate","Main Fire Size","Main Flame
Height","Main Convec. Size","Plume Flow(  1)","Pyrolysis Rate(  1)","Fire Size(  1)","Flame Height(  1)","Convec. Size(
1)","Target Temperature(  1)","Target Flux(  1)","Target Fire Rad.(  1)","Target Surface Rad.(  1)","Target Gas Rad.(
1)","Target Convec.(  1)","Target Temperature(  2)","Target Flux(  2)","Target Fire Rad.(  2)","Target Surface Rad.(
2)","Target Gas Rad.(  2)","Target Convec.(  2)","Target Temperature(  3)","Target Flux(  3)","Target Fire Rad.(
3)","Target Surface Rad.(  3)","Target Gas Rad.(  3)","Target Convec.(  3)"

row 2 (0 seconds):
0.000000E+00,0.300000E+03,0.300000E+03,0.459954E+01,0.209300E-01,0.324148E-20,0.000000E+00,0.000000E+00,0.000000E+00,0.0000
00E+00,0.000000E+00,0.000000E+00,0.000000E+00,0.000000E+00,0.000000E+00,0.000000E+00,0.000000E+00,0.000000E+00,0.300000E+03
,0.459270E+03,0.000000E+00,0.000000E+00,0.000000E+00,0.000000E+00,0.300000E+03,0.459270E+03,0.000000E+00,0.000000E+00,0.000
000E+00,0.000000E+00,0.300000E+03,0.459270E+03,0.000000E+00,0.967660E+02,0.323401E+01,0.000000E+00

row 17 (1800 seconds):
0.180000E+04,0.377000E+03,0.303842E+03,0.723557E+00,0.176378E+03,-.948170E-01,0.436614E+03,0.334626E+03,0.291118E+00,0.4149
38E-02,0.999839E+05,0.110673E+01,0.699887E+05,0.502605E-01,0.310000E-02,0.498795E+04,0.119848E+00,0.349156E+04,0.319643E+03
,0.131802E+04,0.000000E+00,0.000000E+00,0.000000E+00,0.000000E+00,0.367099E+03,0.647807E+03,0.000000E+00,0.000000E+00,0.000
000E+00,0.000000E+00,0.310750E+03,0.863351E+03,0.000000E+00,0.279026E+02,0.747219E+02,-.262447E+01
```

There is a label in quotes and, the compartment or counter (target) in parentheses. The units are the same as in report (SI), and flux is net flux to the surface or target.

5.3 Error Messages

In some (hopefully rare) cases, a simulation will fail to complete. In those cases, an error message provides guidance to the user on possible reasons for the failure. The message will contain an error number which provides a reference to additional information from the table below. Most often, these errors result from improper information in the input data files.

81

Table 49. CFAST Error Messages

Error Number	Error(s) Causing This Message	Possible Fixes
1	Wrong CFAST fire type	The data file for the chosen scenario defines the fire type by number. In a CFAST input file, this number is identified on the line beginning with LFBT. This number must be 0, 1, or 2.
4	No data file selected	The CFAST fire model was executed without an associated data file.
14	Interior node has too many or too few connections	A maximum of three connections to a single node is allowed for each interior connection in a mechanical ventilation system. Revise the test case so that the maximum number of connections to each node is less than or equal to three.
16	Node range exceeded	A maximum of 62 nodes are allowed in a mechanical ventilation systems. Simplify the test case to reduce the total number of nodes in the mechanical ventilation systems.
17	RAD2 – singular matrix	An internal error has occurred calculating radiative heat transfer. This message should not occur.
18	RAD4 – singular matrix	An internal error has occurred calculating radiative heat transfer. This message should not occur.
19	Overwrite, input and index	An internal error has occurred writing the history file to disk. This message should not occur. Revise the test case so that a history file is not generated.

Error Number	Error(s) Causing This Message	Possible Fixes
20	Incorrectly defined object type	Within the object database, each object fire is defined by an object type and additional inputs that depend upon the type of the object. Currently, only three different types are allowed: Unconstrained, Constrained, or Flame Spread. Most typically, this error results from a corrupt object database file. Reinstall the CFAST software to recover the original object database. Note that any changes made to the object database will be lost.
21	Failure of target initialization	Within the CFAST fire model, heat flux to the floor is calculated by defining a position on the floor called a target. An internal error has occurred in the definition of the target. This message should not occur.
22	Error initializing graphics descriptor	The selected input file contains errors in the description of graphical output beginning with the DEVICE line in the input file. This message should not occur. Use a text editor to remove all lines from the line beginning with DEVICE to the end of the file.
24	Invalid detector specification	There is a syntax error in the input file describing one of the detectors included in the scenario.
25		
36	Thermophysical properties file does not exist.	The file THERMAL.DF has been deleted from the CFAST subdirectory. Reinstall the CFAST software to recover the file.
37	Improper definition of thermophysical properties	One of the files THERMAL.DF or THERMAL.NDX has been corrupted and contains invalid information defining thermal properties of materials. Reinstall the FAST / FASTLite software from the CD-ROM to recover the files.

Error Number	Error(s) Causing This Message	Possible Fixes
38	Too many time points in fire definition	The defined fire curve has too many times points. A maximum of 21 points can be defined. Define a new fire curve to replace the existing fire curve in the selected data file.
43	Detector is placed in a non-existent compartment	The detector is placed in a compartment which no longer exists in the scenario. Delete defined detectors.
44	RTI value for a detector is invalid	The RTI value for a detector is outside the range of 1 $m^{1/2}s^{1/2}$ to 400 $m^{1/2}s^{1/2}$. Correct RTI value
45	Location of detector is outside physical dimensions of compartment	The specified location for a detector is outside the defined dimensions for a compartment. Check detector locations.
46	Invalid detector type	Detectors may be either smoke detectors or heat detectors. The fire model has read in a definition for a detector which is neither. Check detector type.

Error Number	Error(s) Causing This Message	Possible Fixes
Less than 0	DASSL error	An error has occurred in the solution of the differential equation set in the fire model. The differential equation solver in CFAST, DASSL, has returned an error message after failing to converge to a solution within acceptable error tolerances. There are many possible causes for this error. Often, changing the size of ventilation openings to the outside allows the simulation to proceed. Providing sufficiently large openings for the first few seconds of a simulation allows the differential equation solver to proceed without effecting the ultimate outcome of the fire. Once the model has initialized to appropriate values, the vents can be closed and the simulation allowed to proceed. Defining an extremely large fire in a small compartment with limited ventilation can cause the differential equation solver to fail. Usually the fire size is orders of magnitude larger than can be supported with available oxygen. Reduce the fire size to allow the simulation to proceed.

6 Concluding Remarks

CFAST is a collection of data, computer programs, and documentation which are used to *simulate* the important time-dependent phenomena describing the character of a compartment fire. The major functions provided include calculation of the buoyancy-driven as well as forced transport of energy and mass through a series of specified compartments and connections (e.g., doors, windows, cracks, ducts), and the resulting temperatures, smoke optical densities, and gas concentrations after accounting for heat transfer to surfaces and dilution by mixing with clean air.

The CFAST model has been subjected to extensive evaluation studies by NIST and others. Although differences between the model and the experiments were evident in these studies, most differences can be explained by limitations of the model or the experiments. Like all predictive models, the best predictions come with a clear understanding of the limitations of the model and of the inputs provided to do the calculations. The user must supply all of the inputs for the model for any particular simulation, and then validate the predictions of the model with experiments to ensure the accuracy of the simulations. Only then can the model be expected to predict the outcome of fire scenarios that are similar to those that have actually been tested.

7 References

[1]   Jones, W.W., Peacock, R.D., Forney, G.P., and Reneke, P.A., "CFAST – Consolidated Model of Fire Growth and Smoke Transport (Version 5), Technical Reference Guide," National Institute of Standards and Technology, Spec. Pub. 1030 (2004).

[2]   Gross, D., "Data Sources for Parameters Used for Predictive Modeling of Fire Growth and Smoke Spread," National Institute of Standards and Technology, NISTIR 85-3223 (1985).

[3]   Jones, W.W. and Peacock, R.D., "Technical Reference Guide for FAST Version 18," National Institute of Standards and Technology, TN 1262 (1989).

[4]   Cooper L.Y. and Forney, G. P., "The Consolidated Compartment Fire Model (CCFM) computer application CCFM-VENTS – part I: Physical reference guide," National Institute of Standards and Technology, NISTIR 4342 (1990).

[5]   Peacock, R. D., Jones, W. W., Forney, G. P., Portier, R. W., Reneke, P. A., Bukowski, R. W., and Klote, J. H., "Update Guide for HAZARD I Version 1.2," National Institute of Standards and Technology, NISTIR 5410 (1994).

[6]   Gross, D., "Data Sources for Parameters Used for Predictive Modeling of Fire Growth and Smoke Spread," National Institute of Standards and Technology, NISTIR 85-3223 (1985).

[7]   Forney, G. and McGrattan, K., "User's Guide for Smokeview Version 4," National Institute of Standards and Technology Special Publication 1017 (2004).

[8]   D. Gross, Data Sources for Parameters Used in Predictive Modeling of Fire Growth and Smoke Spread, National Institute of Standards and Technology IR 85-3223 (1985).

[9]   U.S. Standard Atmosphere, 1976, U.S. Government Printing Office, Washington, D.C., 1976.

[10]   The CRC Handbook of Chemistry and Physics, David Lide, Editor, CRC Press, Boca Raton (1994).

[11]   Forney, G. P. , "Note on Improving Corridor Floor Predictions in a Zone Fire Model," National Institute of Standards and Technology, NISTIR 6046 (1997).

[12]   Klote, J. H., "Fire Experiments of Zoned Smoke Control at the Plaza Hotel in Washington DC," National Institute of Standards and Technology, NISTIR 90-4253 (1990).

[13]   Klote, J.K. and Milke, J.A., Design of Smoke Management Systems, American Society of Heating, Refrigerating and Air-conditioning Engineers, Atlanta, GA (1992).

[14] ASHRAE Handbook HVAC Systems and Equipment, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Atlanta, GA (1992).

[15] Klote, J. H., A Computer Model of Smoke Movement by Air Conditioning Systems, NBSIR 87-3657 (1987).

[16] Huebscher, R. G., "Friction Equivalents for Round, Square, and Rectangular Ducts," ASHVE Transactions (renamed ASHRAE Transactions), Vol. 54, pp 101-144.

[17] Heyt, J. W., and Diaz, J. M., "Pressure Drop in Flat-Oval Spiral Air Duct," ASHRAE Transactions, Col. 81, Part 2, pp 221-230.

[18] Welty, J.R, Wicks, C.E, Wilson, R.E., *Fundamentals of Momentum, Heat and Mass Transfer, 3rd ed.,* John Wiley and Sons, New York, NY, 1984.

[19] Moody, L.F., "Friction "Factors for Pipe Flow," Transactions of ASME, Vol. 66, 1944, p. 671-684.

[20] Moody, L.F., "Friction Factors for Pipe Flow," *Transactions of ASME*, **66** pp 671-684 (1944).

[21] G. W. Mulholland and C. Croarkin, Specific Extinction Coefficient of Flame Generated Smoke, Fire and Materials *24*, 227 (2000).

[22] Jin, T. , "Evaluation of Fire Exit Signs in Smoke," Systems Approach to Fire Safety in Buildings. Volume 2. Session 3. Active Systems Performance and Criteria: Smoke Control, Detection, Sprinklers. Session 4. Passive Systems Performance and Criteria: Combustibles, Fire Resistance. August 29-30, 1979, Tsukuba, Japan, III/73-78 pp, 1979.

[23] Drysdale, D., "An Introduction to Fire Dynamics," John Wiley and Sons, New York, (1985).

[24] Tewarson, A., "Combustion of Methanol in a Horizontal Pool Configuration," Factory Mutual Research Corp., Norwood, MA, Report No. RC78-TP-55 (1978).

[25] McCaffrey, B.J., "Entrainment and Heat Flux of Buoyant Diffusion Flames," National Bureau of Standards (U. S.), NBS IR 82-2473 (1982).

[26] Koseki, H., "Combustion Properties of Large Liquid Pool Fires," *Fire Technology 25*, 241 (1989).

[27] Cleary, T. G. and Quintiere, J. G., "A Framework for Utilizing Fire Property Tests," Natl. Inst. Stand. Technol., NISTIR 4619 (1991).

[28] Babrauskas, V. and Grayson, S. J., "Heat Release in Fires" Elsevier Applied Science, London (1992).

[29]   Evans, D. D., "Sprinkler Fire Suppression Algorithm for HAZARD," Natl. Inst. Stand. Technol., NISTIR 5254 (1993).

[30]   Davis, W. D. And Notarianni, K. A., "NASA Fire Detector Study," Natl. Inst. Stand. Technol., NISTIR 5798 (1996).

[31]   Cooper, L. Y., Fire-Plume-Generated Ceiling Jet Characteristics and Convective Heat Transfer to Ceiling and Wall Surfaces in a Two-Layer Zone-Type Fire Environment, National Institute of Standards and Technology, IR 4705, 57 p. (1991).

[32]   Heskestad, G., "Fire Plumes, Flame Height, and Air Entrainment" in *The SFPE Handbook of Fire Protection Engineering*, 3rd Ed., National Fire Protection Association (2002).

[33]   Peacock, R. D., Jones, W. W., Bukowski, R. W., and Forney, C. L., "Technical Reference Guide for the HAZARD I Fire Hazard Assessment Method, Version 1.1," Natl. Inst. Stand. Technol., Handbook 146 (1991).

Appendix A Databases Used by the Model

The models uses two databases, one for the thermophysical properties needed for conduction in bounding surfaces and targets, and the other for implementing more than a single (main) fire.

A.1 Other Objects

Burning items other than the main fire must be defined in the "other objects" database. The format of the file is

line 1) OBJECT OBJNAM
      OBJNAM(NUMOBJL) = Name of object (up to 8 characters)

line 2) OBJTYP OBJCRI(2,I) OBJCRI(3,I) OBJMAS OBJGMW OBJVT OBJHC
      OBJTYP(NUMOBJL) = Object type
              1 unconstrained burn
              2 constrained burn
              3 flame spread model
      OBJCRI(2,NUMOBJL) = Flux for ignition (W/m^2)
      OBJCRI(3,NUMOBJL) = Surface temperature for ignition (k)
      OBJMAS(NUMOBJL)   = Total mass (kg)
      OBJGMW(NUMOBJL)   = Gram molecular weight
      OBJVT(NUMOBJL)   = Volatilization temperature (k)
      OBJHC(NUMOBJL)   = Heat of combustion (J/kg)

line 3) OBJXYZ(1,I) OBJXYZ(2,I) OBJXYZ(3,I) OBJORT(1,I) OBJORT(2,I) OBJORT(3,I)
      OBJXYZ(1,NUMOBJL) = Panel length (m)
      OBJXYZ(2,NUMOBJL) = Panel height or width (m)
      OBJXYZ(3,NUMOBJL) = Panel thickness (m)
      OBJORT(1,NUMOBJL) = x component of the normal
      OBJORT(2,NUMOBJL) = y component of the normal
      OBJORT(3,NUMOBJL) = z component of the normal

For type 1 and 2 fires line 4) is blank. For type 3 objects line 4 has data specific only to flame spread. The format for a flame spread object follows.

line 4) OBJMINT OBJPHI OBJHGAS OBJQAREA
      OBJMINT = Minimum surface temperature for lateral flame spread (K)
      OBJPHI = Lateral flame spread parameter (J^2/s^2*m^3)

OBJHGAS = Effective heat of gasification (J/kg)
OBJQAREA   = Total heat per unit area (J/m^2)

The effective heat of gasification is the one unusual parameter for the flame spread object. The method given by Brown in *Heat Release in Fires* [28] is to plot the peak heat release rate (HRR) against the radiant flux used for a number of cone calorimeter runs at different fluxes. The slope of the curve that bests fits the data is taken to be the ratio of the effective heat of combustion to the heat of gasification.

line 5) OTIME(1,I) to OTIME(TOTJ,I)

OTIME(NV,NUMOBJL) = Time history (s)

Each OTIME(J,I) represents a point on the objects burn time line where the variables below are defined exactly. CFAST will interpolate values between any two points. TOTJ is the total number of points on the specified time line. CFAST automatically assigns an initial time zero for the objects time line so that there will always be one fewer specified value for the time line than for the history variables below. For type three objects the curves are straight lines to represent the constants that CFAST uses for the flame spread objects.

The following lines are the histories for the individual parameters at each of the OTIME points.

Line 6) OMASS(1,I) to OMASS(TOTJ+1,I)
        OMASS(NV,NUMOBJL) = Pyrolysis rate time history (kg/s)

line 7) OQDOT(1,I) to OQDOT(TOTJ,I)
        OQDOT(NV,NUMOBJL) = Rate of heat release time history (w)

line 8) OAREA(1,I) to OAREA(TOTJ,I)
        OAREA(NV,NUMOBJL) = Area of fire time history (m^2)

line 9) OHIGH(1,I) to OHIGH(TOTJ,I)
        OHIGH(NV,NUMOBJL) = Height of flame time history (m)

line 10) OCO(1,NUMOBJL) to OCO(TOTJ,NUMOBJL)
        OCO(NV,NUMOBJL) = CO/CO2 time history

line 11) OOD(1,NUMOBJL) to OOD(TOTJ,NUMOBJL)
        OOD(NV,NUMOBJL) = OD or soot time history

line 12) OHRC(1,NUMOBJL) to OHRC(TOTJ,NUMOBJL)
        OHCR(NV,NUMOBJL) = H/C time history

line 13) OOC(1,I) to OOC(TOTJ,I)
       OOC(NV,NUMOBJL) = O/C time history

line 14) OMPRODR(1,10,I) to OMPRODR(TOTJ,10,I)
       OMPRODR(NV,10,NUMOBJL) = CT time history

line 15) OMPRODR(1,5,I) to OMPRODR(TOTJ,5,I)
       OMPRODR(NV,5,NUMOBJL) = HCN time history

line 16) OMPRODR(1,6,I) to OMPRODR(TOTJ,6,I)
       OMPRODR(NV,6,NUMOBJL) = HCl time history

A.2 Thermophysical Properties

Thermophysical properties are defined in the database THERMAL.DF, which can be changed by using the THRMF keyword. The relationship is by the name used in specifying the boundary. Any name can be used so long as it is in the thermal database. If a name is used which is not in the database, then CFAST will turn off the conduction calculation, and FAST will stop with an appropriate error message. The form of an entry in the database is:

name conductivity specific heat density thickness emissivity

and the units are:

        name   1 to 8 alphanumeric characters
        conductivity   Watts/meter/Kelvin
        specific heat   Joules/kilogram/Kelvin
        density   kilograms/cubic meter
        thickness   meters
        emissivity   dimensionless.

The first three entries in the default data base are

```
GYPSUM    0.16 900 790 0.016 0.9 0.0063 191.8 0.0587 7476 193 1.021 0.431   Gypsum Board (5/8")
GYP1/2    0.16 900 790 0.013 0.9 0.0063 191.8 0.0587 7476 193 1.021 0.431   Gypsum Board (½")
GYP3/4    0.16 900 790 0.019 0.9 0.0063 191.8 0.0587 7476 193 1.021 0.431   Gypsum Board (3/4")
```

which specifies three different thicknesses of gypsum, whose conductivity is 0.16 W/m K, specific heat is 900 J/k K and density is 790 kg/m^3. The fourth number is the thickness and is represented in the name (normal, 0.013 m and 0.019 m). The fifth is the emissivity, 0.9 in the above example. The next seven parameters deal with adsorption/absorption of hydrogen chloride on the surface. This is discussed in detail in the Technical Reference Guide.

All of the examples shown above represent single layer partitions. The model, however, will handle up to three layers, or slabs, per partition. To specify multiple layers, each property is given one, two or three values separated by slashes. It is important that this be consistent across the whole entry, otherwise CFAST will have an ambiguous situation. The ambiguity is resolved by using the lowest number of layers.

The layers are numbered from the inside to outside. That is, the first material property represents the material on the compartment interior, and the last property is on the outside. There are several examples of this format in the THERMAL.DF file included in this distribution. One example is:

FC1_L003 .15/.03/.22 1500./1042./1047. 790./1.2/128. .016/.24/.019 .9

Since the emissivity applies only to the interior surface, it will always have only a single entry. Other values are ignored. This example represents a three layer assembly taken from Underwriters Laboratory (UL) "Fire Resistance Directory." The code stands for Floor Combustible 1 Hour, and L003 is assigned by UL.

Finally, the HCl coefficients are included for each entry in THERMAL.DF since they pertain to surface effects, and the current algorithm uses a similarity solution for the heat and mass transfer. There are seven coefficients for the HCl absorption. If the first two are zero, the others are not used even though they must be present. These coefficients have only been measured for a few materials. The most commonly used material is:

GYPSUM   [0.16 900. 790. 0.016 0.90] 0.0063 191.8 0.0587 7476. 193 1.021 0.431

Appendix B Initialization Files

B.1 Solver Initialization

These are the defaults for the solver. The name is SOLVER.INI

```
C    ABS PRESSURE TOL, REL PRESSURE TOL, ABS OTHER TOL, REL OTHER TOL
     DATA APTOL, RPTOL, ATOL, RTOL/1.D-5, 1.D-5, 1.D-2, 1.D-2/
C    ABS WALL TOL, REL WALL TOL
     DATA AWTOL, RWTOL, ALGTOL/1.4D-2, 1.4D-2, 1.D-8/
C    ABS HVAC PRESS, REL HVAC PRESS, ABS HVAC TEMP, REL HVAC TEMP
     DATA AHVPTOL,RHVPTOL,AHVTTOL,RHVTTOL/1.D-5,1.D-5,1.D-2,1.D-2/
     DATA (OPTION(J),J=1,16)/
C    OPTIONS FIRE, HFLOW, ENTRAIN, VFLOW, CJET, DOOR-FIRE, CONVEC, RAD,
     +         2,     1,      1,       1,    1,       1,        1,    1,
C       CONDUCT, DEBUG, EXACT ODE,  HCL , MFLOW, KEYBOARD,
     +         1,     0,      1,       1,    1,       0,
C       TYPE OF INITIALIZATION,   MV HEAT LOSS
     +         1,               0/
C    NUMBER OF WALL NODES, FRACTIONS FOR FIRST, MIDDLE AND LAST WALL SLAB
     DATA NWPTS /20/
C    BOUNDARY CONDITION TYPE (1=CONSTANT TEMPERATURE, 2=INSULATED 3=FLUX)
     DATA IWBOUND /1/
C    COMPUTED VALUES FOR BOUNDARY THICKNESS
     DATA (WSPLIT(J),J=1,3)  /0.65, 0.15, 0.20/
C    TURN DEBUGGING OPTIONS OFF - THIS IS NOT CURRENTLY USED
     DATA DEBUG /MXOPT*0/
C    MAXIMUM STEP SIZE¹, MAXIMUM FIRST STEP: IF NEGATIVE THEN SOLVER WILL DECIDE
     DATA STPMAX /-1.0/, DASSLFTS/-1.0/
```

B.2 Configuration File

The default configuration file is HV5.CF. These entries are fixed format, and must be set as shown.

header (a6): $$CF$$
version, color table: 2311 15 14 15 11 15 15 0 1 1 0 12 6 2 10 13 4 16 16 1T
file name for the thermophysical properties (a60): THERMAL.DF (default is THERMAL.DAT)
restart file (a60): BURN.DAT
geometric database (a60): GEOMETRY.DAT
multiple objects database (a60): OBJECTS.DF (default is OBJECTS.DAT)
partition data base (a60): PARTITIN.DAT
data base path (a64): \NIST\CFAST511\BIN (defaults to current directory if blank)
data directory (a64): \NIST\CFAST511\BIN\DATA (defaults to current directory if blank)
default graphics display (for pc displays) (a2,i1): IV3

---

[1] This value may be modified with the keyword STPMAX. Normally, it should be left at the default value to allow the solver to chose the step size automatically.

Note that the version number is the current verion, 511 offset by 1800 which is the transition version of FAST.

B.3 Setting Options for the Solver (DASSL)

The following table documents components of the OPTION array in CFAST. This array is defined in the FORTRAN subroutine INITSOLN.sor and is over-ridden by the data file SOLVER.INI. The OPTION array is used to turn on and off various sub-models in CFAST for debugging purposes. If any component of OPTION is set to zero then the sub-model or CFAST using that component is not used during the current calculation.

Option	index	default	Description (0 is off)
FFIRE	1	2	then the "CFAST" fire is used. (1 is not used)
FHFLOW	2	1	Horizontal flow.
FENTRAIN	3	1	Entrainment.
FVFLOW	4	1	Vertical flow.
FCJET	5	2	Ceiling jet (1 is not used)
FDFIRE	6	1	Door jet fires.
FCONVEC	7	1	Convection.
FRAD	8	1	1 - call ABSORB for all rooms (use species) 2 - call ABSORB only for the fire room (use 0.5, 0.01 otherwise) 3 - the gas in the fire room is black (otherwise use 0.5, 0.01) 4 - Use 0.5 and 0.01 (used in 2.n)
FCONDUC	9	1	not used
FDEBUG	10	0	1, 2 and 3 - produce levels of debug print
FODE	11	1	0 - don't use dP/dt in the solution of T and other variables 1 - include dP/dt for V, T, and so on
FHCL	12	1	model HCL. deposition
FMVENT	13	1	Include mechanical ventilation
FKEYVAL	14	1	not used

FPSTEADY	15	1	1 - pressures are initialized so that dP/dt=0. 2 - pressures and upper layer temperatures are initialized so that dP/dt=0 and dTU/dt=0.
FHVLOSS	16	0	Heat loss to mechanical ventilation shafts is modeled.
FMODJAC	17	1	Structure of the Jacobian is exploited to produce more efficient (ie faster) calcuations
FPDASSL	18	0	Produce debug print
FOXYGEN	19	0	1 - species oxygen is solved for by DASSL.
FBDTECT	20	0	not used
FBTOBJ	21	0	Objects are "backtracked" after burning starts