



CONTAM93 User Manual

George N. Walton

Building and Fire Research Laboratory
Gaithersburg, Maryland 20899

QC
100
.U56
1994
#5385



United States Department of Commerce
Technology Administration
National Institute of Standards and Technology

CONTAM93
User Manual

George N. Walton

March 1994
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899



U.S. Department of Commerce
Ronald H. Brown, *Secretary*
Mary L. Good, *Under Secretary for Technology*
National Institute of Standards and Technology
Arati Prabhakar, *Director*

ABSTRACT

This manual describes the use of two computer programs for analyzing the air movement and indoor air quality in multizone buildings. The first program CONTAM is used to create and edit the building description including data for all features relating to airflow or to the generation and removal of contaminants. It uses a graphic interface to establish the spatial relationship of these features. These data along with weather data are used by the program CONTAMX to calculate the airflows and dynamic levels of indoor contaminants. The results of the calculation may be reviewed graphically in CONTAM and printable files may be generated. Together these two programs are called CONTAM93.

DISCLAIMER

The programs described in this report are furnished by the government and are accepted and used by any recipient with the express understanding that the United States Government makes no warranty, expressed or implied, concerning the accuracy, completeness, reliability, usability, or suitability for any particular purpose of the information and data contained in this program or furnished in connection therewith, and the United States shall be under no liability whatsoever to any person by reason of any use made thereof. These programs belong to the government. Therefore, the recipient further agrees not to assert any proprietary rights therein or to represent these programs to anyone as other than government programs. Any commercial products or tradenames mentioned in this report are included for informational purposes only, and do not constitute an endorsement or recommendation by the National Institute of Standards and Technology.

TABLE OF CONTENTS

1. INTRODUCTION	1
1.1 Background	1
1.2 Purpose	1
1.3 Scope of report	1
2. GENERAL DESCRIPTION	2
3. SIMULATION FEATURES	4
3.1 Contaminant Analysis	4
3.2 Airflow Analysis	6
4. COMPUTING ENVIRONMENT	13
4.1 Requirements	13
4.2 Installation	13
4.3 CONTAM93 Files	14
4.4 Running CONTAM93	15
5. FUTURE DEVELOPMENTS	16
6. ACKNOWLEDGMENTS	17
7. REFERENCES	17
APPENDIX A: General Description	A.1
APPENDIX B: The CONTAM User Interface	B.1
APPENDIX C: CONTAM Data Entry Screens and Help Messages	C.1
APPENDIX D: CONTAM File Formats	D.1

1. INTRODUCTION

1.1 Background

The National Institute of Standards and Technology (NIST) has over the past several years developed a series of public domain computer programs for calculating air flow and contaminant dispersal in multizone buildings. The most recent of these is NBSAVIS/CONTAM88 (Grot, 1991).

There are several applications for building airflow calculations. The earliest such NIST (then NBS, the National Bureau of Standards) program was ASCOS (Analysis of Smoke Control Systems) (Klote, 1981, 1983). Another program, TARP (Thermal Analysis Research Program) (Walton, 1983, 1984), used multizone airflow calculations to estimate the portion of building thermal load due to infiltration and perform a simple contaminant migration analysis. Programs were developed specifically for the study of contaminant migrations (CONTAM86 and CONTAM87, Axley, 1986, 1988). Improvements in the algorithm for computing airflows were described in the AIRNET program (Walton, 1989).

All of these programs take the multizone network approach to airflow analysis. That is, the building is treated as a collection of zones connected by airflow paths. These zones may represent groups of rooms, individual rooms, or even portions of rooms, as well as shafts and portions of the building air handling system. Within each zone the temperature and contaminant concentration is considered to be uniform. The airflow paths include doorways, small cracks in the building envelope, and a simple model of the air handling system. This approach is a compromise between the single-zone building model, which is appropriate only for the simplest of buildings, and computational fluid dynamics models, which are still too time consuming for use on a regular basis, especially for large buildings.

Use of these programs has indicated that the computational algorithm in ASCOS needs to be replaced by a faster and more robust algorithm such as used by AIRNET, while the AIRNET user interface which was designed for research purposes needs to be replaced by a much more easily used interface. It was decided to combine the best available algorithms with a graphic interface in a program usable on commonly available small computers. This program is called CONTAM93.

1.2 Purpose

The purpose of this effort was to develop an easily used contaminant analysis program using a graphic interface with the contaminant simulation capabilities of CONTAM88 and the airflow analysis method of AIRNET.

1.3 Scope of report

The following sections of this report contain a general description of CONTAM93, a description of the contaminant dispersal and airflow calculation algorithms, and some necessary details on the computational environment. The appendices give a detailed description of the user interface.

2. GENERAL DESCRIPTION

CONTAM93 is an easily used contaminant analysis program with the contaminant simulation capabilities of CONTAM88 (Grot, 1991) and the airflow analysis method of AIRNET (Walton, 1989).

CONTAM93 requires a 286 class (or higher) PC compatible computer with math coprocessor, VGA graphics, and MS-DOS. CONTAM93 consists of two programs: CONTAM and CONTAMX. CONTAMX is a non-interactive program which computes the airflows and contaminant dispersal. CONTAM is an interactive program for processing the CONTAMX input and output files. Both CONTAM and CONTAMX operate under the 640K-byte memory limit of MS-DOS, but this still leaves room for simulating some very large buildings.

CONTAM93 is designed to simplify the data input and analysis processes for a multi-zone airflow and contaminant dispersal simulation. It is still up to the user to determine what processes are important and how the complexities of a building can be abstracted into the mathematically computable model used by CONTAM93. A lengthy data gathering process may be required.

The user does not directly access the data files describing the building. All access is done through the CONTAM program and its graphic interface. The main portion of the computer screen will be occupied by the "SketchPad" or a data entry screen as described below. The options area to the right indicates possible actions that can be taken by pressing the keys indicated. The status area at the bottom of the screen displays information about the current object on the SketchPad, error messages, and questions. The screen interface is explained in detail in the appendices.

The description of the building is created (or modified) in the SketchPad. The SketchPad consists of an array of small cells into which the user places various symbols representing building features relevant to the calculation of contaminant dispersal. This produces a simple illustration which has been chosen intentionally to represent the simplicity of the underlying mathematical model. The SketchPad is used to establish the geometric relationships of the relevant building features. It is not intended to produce a scale drawing of the building. It is used to create a simplified model where the walls, zones, and air flow paths are topologically similar to the actual building. Engineering judgement must be used to define this model of the building. The SketchPad allows the entry and display of the data in an intuitive manner. The SketchPad will bring up various data entry screens needed to define the mathematical characteristics of the various building features.

Building data is organized by "levels" with data entry beginning at the lowest level. A level would typically be a building floor, but a suspended ceiling acting as a return air plenum or a raised floor acting as a supply plenum may also be treated as a level leading to multiple levels per floor. Define the walls first. "Walls" are used to separate regions of uniform air, called zones, on a given level. Such walls include the building envelope and internal partitions with a significant resistance to air flow. Walls must be drawn to create enclosed regions (zones). There is a set of implicit walls (actually floors) separating the zones on different levels. Identify the zones next. A "zone" indicates a volume of air with uniform temperature and contaminant concentration. Zones are defined on the SketchPad as a region surrounded by walls. Every zone must be identified. A default ambient zone surrounds the building. An airflow path indicates some building feature by which air can move from one zone to another. Such features include cracks in the building envelope, open doorways, and exhaust fans. Path symbols placed on the walls are used to represent openings between zones or to ambient; any other placement represents an opening in the floor to the zone on the level below. Contaminant source (or sink) symbols may be placed in any zone. These represent any feature (within the list of available models) which produces or removes a contaminant. A simple model of an air handling system is available with supply and return point

symbols placed within the appropriate zones.

Figure 1 shows one level of a building with lines representing the walls that define the zones, and diamond shaped symbols representing the airflow paths between zones. As you move the cursor across zone and path symbols, summary data is displayed on the screen. Pressing the enter key while at a zone, path, or contaminant symbol gives access to the complete data and permits you to change the data. Other keys allow you to copy, move, or delete various symbols. A model of a multi-level building is built one level at a time. Since levels in tall buildings tend to be similar, the user can create a level by copying a lower level and editing the SketchPad display.

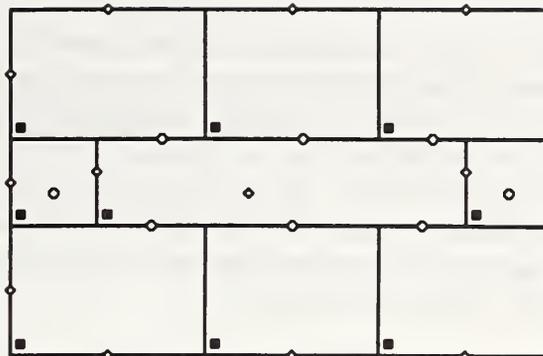


Figure 1. SketchPad Example

In addition to the relationships established on the SketchPad, a great deal of numeric data is required to define the mathematical model for the simulation. This data is prepared in the data entry screens. A data entry screen consists of one or more data entry fields and accompanying text. The data entry fields are generally organized with the most important data toward the top of the screen. A more detailed explanation of the data entries is available from the on-line help procedure. Default values are often provided. Reminders about how to use the data entry screen are provided in the options window to the right. There are several different types of data entry fields. They allow an action to be taken, a choice made, a name given, or a numeric value specified. Data entries representing physical values also have the units displayed in brackets, e.g. [m²]. The user can change the units. In order to minimize the amount of numerical data entered, data representing a single building feature, such as a doorway, is grouped together and given a name by the user. When that feature is needed at another place in the building, it can be referred to by name instead of re-entering all the numeric data.

Once the building description is complete, the user may select one of several simulation modes to be done by CONTAMX. Steady, transient (up to 24 hours), and cyclic (24-hour steady-periodic) contaminant simulations are possible. Steady or dynamic (transient / cyclic) calculation of the airflows may be done. The results of the simulation can be graphically viewed with CONTAM using the SketchPad display. The display shows the pressure drops and flow rates for the airflow paths. The zone contaminant concentrations can be displayed by moving the cursor to a zone symbol. For transient simulations, concentrations, each normalized for the peak value, are presented graphically. The user may also create files which can be printed or used in other applications. The zone and path file uses a format similar to the one used by the ASCOS program. The contaminant data is printed in two different formats based on whether a steady or transient simulation was done.

The building descriptions created with CONTAM are saved in project data files. These files must be saved to keep a record of what was simulated. The various reports provide only the results of simulation, not the building description. A research or design project will usually involve several different project files describing different conditions from which comparisons and choices will be made.

3. SIMULATION METHODS

3.1 Contaminant Analysis

The CONTAM93 contaminant dispersal model is an implementation of Axley's (1987, 1988) methods. He states: "The central concern of indoor air quality analysis is the prediction of airborne contaminant dispersal in buildings. Airborne contaminants disperse throughout buildings in a complex manner that depends on the nature of air movements in-to, out-of, and within the building system; the influence of the heating, ventilating, and air-conditioning (HVAC) systems; the possibility of removal, by filtration, or contribution, by generation, of contaminants; and the possibility of chemical reaction, radio-chemical decay, settling, or sorption of contaminants. In indoor air quality analysis we seek to comprehensively model all of these phenomena."

The mass of contaminant α in zone i is

$$m_{\alpha,i} = m_i \cdot C_{\alpha,i} \quad (1)$$

where m_i is the mass of air in zone i and $C_{\alpha,i}$ is the concentration mass fraction of α . As previously stated, a zone is a region of uniform contaminant concentration which may correspond to a single room, a portion of a room, or several well-coupled rooms. Selection of the zones is a matter of engineering judgement. In CONTAM93 the contaminant concentration must be low enough so as not to significantly change the density of the air-contaminant mixture.

3.1.1 Solution of the Equations

Contaminant is removed from the zone i by:

- outward airflows at the rate $\sum_j w_{i,j} C_{\alpha,i}$ where $w_{i,j}$ rate of air flow from zone i to zone j ,
- removal at the rate $C_{\alpha,i} R_{\alpha,i}$ where $R_{\alpha,i}$ is a removal coefficient, and
- first-order chemical reactions with other contaminants $C_{\beta,i}$ at the rate $m_i \sum_{\beta} \kappa_{\alpha,\beta} C_{\beta,i}$ where $\kappa_{\alpha,\beta}$ is the kinetic reaction coefficient in zone i between species α and β . (Sign convention: positive κ for generation and negative κ for removal)

Contaminant is added to the zone by:

- inward airflows at the rate $\sum_j (1-\eta_{\alpha,j,i}) w_{j,i} C_{\alpha,j}$ where $\eta_{\alpha,j,i}$ is the filter efficiency in the path from zone j to zone i ,
- generation at the rate $G_{\alpha,i}$, and
- reactions of other contaminants.

Conservation of contaminant mass for each species (and assuming trace dispersal - $m_{\alpha,i} \ll m_i$) produces the following basic equation for contaminant dispersal in a building:

$$\frac{dm_{\alpha,i}}{dt} = -R_{\alpha,i} C_{\alpha,i} - \sum_j w_{i,j} C_{\alpha,i} + \sum_j w_{j,i} (1 - \eta_{\alpha,j,i}) C_{\alpha,j} + m_i \sum_{\beta} \kappa_{\alpha,\beta} C_{\beta,i} + G_{\alpha,i} \quad (2)$$

This differential equation is approximated by a difference equation:

$$m_{\alpha,i}^* = m_{\alpha,i} + \Delta t \left[\sum_j w_{j,i} (1 - \eta_{\alpha,j,i}) C_{\alpha,j} + m_i \sum_{\beta} \kappa_{\alpha,\beta} C_{\beta,i} + G_{\alpha,i} - (R_{\alpha,i} + \sum_j w_{i,j}) C_{\alpha,i} \right] \quad (3)$$

where the * indicates a value at time $t + \Delta t$.

Instead of the generalized trapezoidal method used by Axley, CONTAM93 uses a fully implicit numerical approximation to solve equation (3):

$$[m_i + \Delta t (R_{\alpha,i} + \sum_j w_{i,j})] C_{\alpha,i}^* - \Delta t \sum_j (1 - \eta_{\alpha,j,i}) w_{j,i} C_{\alpha,j}^* - m_i \Delta t \sum_{\beta} \kappa_{\alpha,\beta} C_{\beta,i}^* \approx m_i C_{\alpha,i} + G_{\alpha,i} \Delta t \quad (4)$$

m_i is considered to be constant. Some special cases are handled by variants of the above equation. If zone i is specified as having no mass (e.g. in the ductwork), equation (4) becomes:

$$(R_{\alpha,i} + \sum_j w_{i,j}) C_{\alpha,i}^* - \sum_j (1 - \eta_{\alpha,j,i}) w_{j,i} C_{\alpha,j}^* \approx G_{\alpha,i} \quad (5)$$

Note that the kinetic reactions cannot be considered in this case. If a steady state solution is desired, equation (4) becomes:

$$(R_{\alpha,i} + \sum_j w_{i,j}) C_{\alpha,i} - \sum_j (1 - \eta_{\alpha,j,i}) w_{j,i} C_{\alpha,j} - m_i \sum_{\beta} \kappa_{\alpha,\beta} C_{\beta,i} \approx G_{\alpha,i} \quad (6)$$

If the mass flows and $R_{\alpha,i}$ should happen to be zero in equations (5) or (6), they cannot be solved. This case is handled by the simple expedient of letting $C_{\alpha,i}$ remain at its default value. These equations must be solved simultaneously for all zones and contaminants.

There are two major limitations in the equations presented above. First is constant zone air mass. This is consistent with the zone temperatures being constant and contaminant mass fractions being low. Second is the use of linear analysis which limits the kinds of kinetic reactions which can be modeled.

3.1.2 Contaminant Source/Sink Elements

CONTAM93 allows the user to define the generation, G_{α} , and removal, R_{α} , coefficients for some simple cases. The general source/sink model uses the following equation:

$$S_{\alpha} = G_{\alpha} - R_{\alpha} \cdot C_{\alpha} \quad (7)$$

where S_{α} is called the contaminant α "source strength". The CONTAM internal units for the terms in equation (7) are: C_{α} [$\text{kg}_{\alpha} / \text{kg}_{\text{air}}$], G_{α} , S_{α} [$\text{kg}_{\alpha} / \text{s}$], and R_{α} [$\text{kg}_{\text{air}} / \text{s}$]. The user may express these values in a large number of units with automatic conversion to the internal values.

For a room air filtering device, $G = 0.0$ and $R = f \cdot e$ where f is the flow rate of the room air passing through the filter and e is the single pass removal efficiency of the device.

The pressure source/sink model is intended to model contaminant sources which are governed by the inside-outside pressure difference, such as radon or soil gas entry into a basement. In this case the source equation is:

$$S_{\alpha} = G_{\alpha} \cdot (P_{\text{ambc}} - P_i)^n \quad (8)$$

For volatile organic compounds the source equation is sometimes expressed in the form

$$S_{\alpha} = G_{\alpha} \left(1 - \frac{C_{\alpha}}{C_{\text{cutoff}}} \right) \quad (9)$$

where C_{cutoff} is the cutoff concentration at which emission ceases.

3.2 Airflow Analysis

Over the years many methods have been developed to compute the building airflows which are necessary for the contaminant analysis. Feustel and Dieris (1992) report 50 different computer programs for multizone airflow analysis. Note that "zones" go by many other names in these programs, e.g., nodes, cells, and rooms are common alternatives. The airflow calculations in CONTAM93 are based on the algorithms developed in AIRNET (Walton, 1989).

3.2.1 Basic Equations

The air flow rate from zone j to zone i , $w_{j,i}$ [kg/s], is some function of the pressure drop along the flow path, $P_j - P_i$:

$$w_{j,i} = f(P_j - P_i) \quad (10)$$

The mass of air, m_i [kg], in zone i is given by

$$m_i = \rho_i V_i = \frac{P_i V_i}{RT_i} \quad (11)$$

where

- V_i = zone volume [m³],
- P_i = zone pressure [Pa],
- T_i = zone temperature [K], and
- $R = 287.055$ [J/kgK] (gas constant for air).

For a transient solution the principle of conservation of mass states that:

$$\frac{\partial m_i}{\partial t} = \rho_i \frac{\partial V_i}{\partial t} + V_i \frac{\partial \rho_i}{\partial t} = \sum_j w_{j,i} + F_i \quad (12)$$

where

- m_i = mass of air in zone i ,
- $w_{j,i}$ = airflow rate [kg/s] between zones j and zone i : positive values indicate flows from j to i and negative values indicate flows from i to j , and
- F_i = non-flow processes that could add or remove significant quantities of air from the zone.

Such non-flow processes are not considered in CONTAM93 and flows are evaluated by assuming quasi-steady conditions leading to:

$$\sum_j w_{j,i} = 0 \quad (13)$$

3.2.2 Solving the Equations

The steady-state airflow analysis for multiple zones requires the simultaneous solution of equation (13) for all zones. Since the function in equation (10) may be, and usually is, nonlinear, a method is needed for the solution of simultaneous nonlinear algebraic equations. The Newton-Raphson (N-R) method (Conte & de Boor, 1972, p 86) solves the nonlinear problem by an iteration of the solutions of linear equations. In the N-R method a new estimate of the vector of all zone pressures, $\{P\}^*$, is computed from the current estimate of pressures, $\{P\}$, by

$$\{P\}^* = \{P\} - \{C\} \quad (14)$$

where the correction vector, $\{C\}$, is computed by the matrix relationship

$$[J] \{C\} = \{B\} \quad (15)$$

where $\{B\}$ is a column vector with each element given by

$$B_i = \sum_j w_{j,i} \quad (16)$$

and $[J]$ is the square (i.e. N by N for a network of N zones) Jacobian matrix whose elements are given by

$$J_{i,j} = \sum \frac{\partial w_{j,i}}{\partial P_j} \quad (17)$$

In equations (16) and (17) $w_{j,i}$ and $\partial w_{j,i}/\partial P_m$ are evaluated using the current estimate of pressure $\{P\}$. The CONTAM program contains subroutines for each airflow element which return the mass flow rates and the partial derivative values for a given pressure difference input.

Equation (15) represents a set of linear equations which must be set up and solved for each iteration until a convergent solution of the set of zone pressures is achieved. In its full form $[J]$ requires computer memory for N^2 values, and a standard Gauss elimination solution has execution time proportional to N^3 . Sparse matrix methods can be used to reduce both the storage and execution time requirements. A skyline solution process following the method of Dhatt (1984, pp. 282-192) was chosen. This method can be used to solve equations with symmetric or nonsymmetric matrices. It stores no zero values above the highest nonzero element in the columns above the diagonal and no zero values to the left of the first nonzero value in each row below the diagonal. In this case the Jacobian matrix is symmetric.

Analysis of the element models will show that

$$|J_{i,i}| = \sum_{j \neq i} |J_{i,j}| \quad (18)$$

This condition allows a solution without pivoting, although scaling may be useful. Note that the degree of sparsity of the Jacobian matrix after factoring is dependent on the ordering of the zones. Ordering can be improved by various algorithms or rules-of-thumb. In AIRNET it was easy to define an airflow network which has no unique solution. The CONTAM93 graphic input makes it difficult to incorrectly connect the airflow elements in the network.

CONTAM allows zones with either known or unknown pressures. The constant pressure zones are included in the system of equations and equation (15) is processed so as to not change those zone pressures. This gives flexibility in defining the airflow network while maintaining the symmetric set of equations. A sufficient condition for the Jacobian to be nonsingular (Axley, 1987) is that all of the unknown pressure zones be linked by pressure dependent flow paths to (a) constant pressure zone(s). In CONTAM the ambient (or outside) air is treated as a constant pressure zone. In CONTAM93 ambient zone pressure is set to zero causing the computed zone pressures to be values relative to the true ambient pressure and helping to maintain numerical significance in calculating ΔP .

Conservation of mass at each zone provides the convergence criterion for the N-R iterations. That is, if $\sum w_{j,i} = 0$ for all zones for the current system pressure estimate, the solution has converged. The number of iterations can be reduced and sufficient accuracy attained by testing for relative convergence at each zone:

$$\left(\frac{|\sum_j w_{j,i}|}{\sum_j |w_{j,i}|} \right) < \epsilon \quad (19)$$

with a test ($\sum |w_{j,i}| < \epsilon_1$) to prevent division by zero. The magnitude of ϵ can be established by

considering the use of the calculated airflows, such as in an energy balance. In any case, round-off errors may prevent perfect convergence ($\epsilon = 0$).

Numerical tests of the N-R method solution indicated occasional instances of very slow convergence as the iterations almost oscillate between two different sets of values. In AIRNET, this was handled by a Steffensen acceleration process. More recent tests by the author and by Wray (1993) indicate that the use of a simpler constant under-relaxation coefficient produces a faster, reliable convergence acceleration process. Equation (14) for the iteration process becomes

$$\{P\}^* = \{P\} - \omega \{C\} \quad (20)$$

where ω is the relaxation coefficient. A relaxation coefficient of 0.75 has been found to be usable for a broad range of airflow networks. This value is not a true optimum but appears to work quite well without the computational cost of finding the theoretically optimum value.

When convergence is progressing rapidly, under-relaxation ($\omega < 1$) slows convergence compared to no relaxation. To prevent this a global convergence value is computed:

$$\gamma = (\sum_i |\sum_j w_{j,i}|) / (\sum_i \sum_j |w_{j,i}|) \quad (21)$$

When $\gamma^* < \alpha \gamma$, ω is set to 1. Currently CONTAM uses $\alpha = 30\%$. This often reduces the number of iterations.

Newton's method requires an initial set of values for the zone pressures. These may be obtained by including in each airflow element model a linear approximation relating the flow to the pressure drop:

$$w_{j,i} = c_{j,i} + b_{j,i} (P_j - P_i) \quad (22)$$

Conservation of mass at each zone leads to a set of linear equations of the form

$$[A] \{P\} = \{B\} \quad (23)$$

Matrix [A] in equation (23) has the same sparsity pattern as [J] in equation (15) allowing use of the same sparse matrix solution process for both equations. This initialization handles stack effects very well and tends to establish the proper directions for the flows. The linear approximation is conveniently provided by the laminar regime of the element models use by CONTAM, but it also may be provided by a secant approximation to the actual nonlinear behavior. When solving a set of similar problems, as when approximating a transient solution by successive steady-state solutions, it tends to be preferable to use the previous solution for the zone pressures as the initial values for the new problem.

CONTAM93 maintains the same equation ordering for the contaminant equations as is used for airflows except that for each zone there is an equation for each contaminant species. These simultaneous equations are solved by the non-symmetric skyline method.

3.2.3 Airflow Elements

Flow within each airflow element is assumed to be governed by Bernoulli's equation:

$$\Delta P = \left(P_1 + \frac{\rho V_1^2}{2} \right) - \left(P_2 + \frac{\rho V_2^2}{2} \right) + \rho g(z_1 - z_2) \quad (24)$$

where

- ΔP = total pressure drop between points 1 and 2
- P_1, P_2 = entry and exit static pressures
- V_1, V_2 = entry and exit velocities
- ρ = air density
- g = acceleration of gravity (9.81 m/s²)
- z_1, z_2 = entry and exit elevations.

The following parameters apply to the zones: pressure, temperature (to compute density and viscosity), and elevation. The zone elevation values are used to determine stack effect pressures. When the zone represents a room, the airflow elements may connect with the room at other than its reference elevation. The hydrostatic equation is used to relate the pressure difference across a flow element to the elevations of the element ends and the zone elevations, assuming the air in the room is at constant temperature. Pressure terms can be rearranged and a possible wind pressure for building envelope openings added to give

$$\Delta P = P_j - P_i + P_s + P_w \quad (25)$$

where

- P_i, P_j = total pressures at zones i and j
- P_s = pressure difference due to density and elevation differences, and
- P_w = pressure difference due to wind.

Equation (25) establishes a sign convention for direction of flow: positive is from zone j to zone i. Since the airflow elements will be described by a relationship of the form $w = f(\Delta P)$, the partial derivatives needed for [J] in equation (17) are related by $\partial w / \partial P_j = -\partial w / \partial P_i$ which establishes the relation in equation (18).

Most infiltration models are based on the following empirical (powerlaw) relationship between the flow and the pressure difference across a crack or opening in the building envelope:

$$w_{j,i} = C\sqrt{\rho_n} (\Delta P)^n \quad (\Delta P > 0) \quad (26a)$$

or

$$w_{j,i} = -C\sqrt{\rho_n} (-\Delta P)^n \quad (\Delta P < 0) \quad (26b)$$

where

- $w_{j,i}$ = mass flow rate of air through the element from zone j to zone i,
- C = flow coefficient,
- ρ = air density of zone j or i,
- ΔP = total pressure loss across the element ($P_j - P_i$), and
- n = flow exponent.

Theoretically, the value of the flow exponent should lie between 0.5 and 1.0. Large openings are characterized by values very close to 0.5, while values near 0.65 have been found for small crack-like openings. The form of equations (26a,b) is suggested by the orifice equation:

$$Q = C_d A \sqrt{\frac{2 \Delta P}{\rho}} \quad (27)$$

where

- Q = volumetric flow ($Q = w/\rho$),
- C_d = discharge coefficient, and
- A = orifice opening area.

Equation (26) should be considered a correlation rather than a physical law. It can be used with the element leakage area formulation which has been used to characterize openings for infiltrations calculations (ASHRAE, 1993, p 23.15). The author has used it to describe flows through ducts to an accuracy of about 2% over a range of flow rates that vary by a factor of four such as might be found in a VAV system.

The primary advantage of equations (26a,b) for describing airflow components is the simple calculation of the partial derivatives for the Newton's method solution of the simultaneous equations:

$$\frac{\partial w_{j,i}}{\partial P_j} = \frac{n w_{j,i}}{\Delta P} \quad \text{and} \quad \frac{\partial w_{j,i}}{\partial P_i} = \frac{-n w_{j,i}}{\Delta P} \quad (28)$$

However, there is also a problem with equations (28): the derivatives become unbounded as the pressure drop (and the flow) go to zero. A simple way to avoid this problem is suggested by what physically happens at low flow rates: the physical character of the flow (and the form of the equation) changes. It goes from turbulent to laminar. Equation (26) can be replaced by

$$w = \frac{K \rho \Delta P}{\mu} \quad (29)$$

where K = flow coefficient, and μ = viscosity.

The partial derivatives are simple constants:

$$\frac{\partial w_{j,i}}{\partial P_j} = \frac{K \rho}{\mu} \quad \text{and} \quad \frac{\partial w_{j,i}}{\partial P_i} = \frac{-K \rho}{\mu} \quad (30)$$

The origin of this laminar relationship is shown by the duct equations in the next section. This technique has been independently discovered and used by several researchers (Axley, 1987; Isaacs, 1980). Although there is physical reason for using equation (29) at low pressure drops, its purpose here is to assure convergence of the equations when ΔP approaches zero for one of the many flow paths in a complex network, instead of accurately representing airflows which are too small to be of interest.

The CONTAM function for powerlaw elements calculates flows using both the laminar and the turbulent models, equations (26) and (29), and selects the method giving the smaller magnitude flow. There is a discontinuity in the derivative of the $w(\Delta P)$ curve where the two equations intersect. This discontinuity is a violation of one of the sufficient conditions for convergence of Newton's method (Conte & de Boor, 1972, p 86). However, numerical tests conducted by the author for flows at that point using a small airflow network have shown no convergence problem.

CONTAM93 also includes the quadratic airflow model, the 2-way flow doorway model, and the performance curve fan model of AIRNET (Walton, 1989).

3.2.4 Wind Pressure

Wind pressure can be a significant driving force for infiltration. It is a complicated function of wind speed, direction, building surface & surroundings. See Grosso (1992) for a detailed review. In CONTAM93 the wind pressure is defined as

$$P_w = P_b - P_a \quad (31)$$

where

- P_b = the pressure on the building surface, and
- P_a = atmospheric pressure at the same position.

Wind pressure is computed from

$$P_w = P_{dyn} C_p C_\theta \quad (32)$$

where

- P_{dyn} = dynamic pressure of the wind at some reference elevation (one-half the air density time the wind velocity squared),
- C_p = a pressure coefficient accounting building/terrain effects, and
- C_θ = a pressure coefficient accounting for relative wind direction.

The relative wind direction is given by

$$\theta = \theta_w - \theta_s \quad (33)$$

where

- θ_w = wind azimuth angle (N = 0, E = 90, etc.), and
- θ_s = surface azimuth angle.

A rather general method of specifying wind pressures has been implemented in CONTAM93, but it does require a significant amount of effort from the user. P_{dyn} is determined directly from the wind velocity; any elevation adjustment must be made by the user in setting the velocity. C_p is a constant value entered by the user for each building envelope airflow path. C_θ is a user specified profile for the wind direction effect which is usually normalized so that $C_\theta(\theta=0) = 1$.

3.2.5 Development of the Airflow Calculations

The method described above for the calculation of multi-zone airflows is the result of a long evolutionary process. In this section the author wishes to describe how it evolved and acknowledge the contributors. It is hoped that this material will be useful to other researchers in indicating other sources and some unexplored areas where further work might be done.

Work in three different areas (hydraulics, infiltration, and smoke spread) has tended to progress independently without much exchange of ideas. One of the earliest multi-zone airflow programs was developed at NRC Canada (Sander, 1974). Work at NIST (then NBS, the National Bureau of Standards) produced ASCOS (Analysis of Smoke Control Systems) (Klote, 1981, 1983) and TARP (Thermal Analysis Research Program) (Walton, 1983, 1984). A similar computation problem exists for hydraulic

pipng networks with solution techniques dating back to Cross (1934). The difficulty in solving these simultaneous nonlinear algebraic equations has led to a large number of papers reporting various methods. Wood and Rayes (1981) tested several methods for solving the pipe network problem. They found the "sequential node method" similar to that used in ASCOS often failed to converge to a solution, and the "simultaneous node method" similar to that used in the NRC program sometimes failed to converge.

Not included in the tests by Wood and Rayes were Newton-Raphson (N-R) methods which date back to at least 1963 (Martin and Peters, 1963). These would also be classified as "simultaneous node" methods. Lam and Wolla (1972) summarize the problems of the N-R method (problems in getting the initial guess solution, difficulty in determining the Jacobian derivatives, and the N^3 cost of solving the equations) and present their improvements.

Work proceeded independently at NIST on improving the Newton-Raphson method from TARP. Several key improvements were made based on ideas from James Axley (1987) (then at NIST, now at MIT). These include the idea of assembling equations for finite element modeling which led directly to the idea of assembling airflow elements into a network. Another important idea was the "skyline method" sparse matrix solver which greatly improves execution time for larger problems. The skyline method may not be optimum but it is very easy to implement and is never worse than a full matrix solution. It is especially appropriate for calculating airflows in tall buildings. Jim also noted that the infinite derivative at zero pressure difference can be handled by using a linear equation at that point. This idea follows from what physically happens in conduit flow. (This idea was independently developed earlier by Welty (see note by Isaacs & Mills, 1982). This linearization also gives a reasonable set of equations for making the initial estimate of nodal pressures. The increasing power of personal computers and some very useful features of the C programming language (such as memory allocation, data structuring, and pointers to data and functions, together with ANSI standardization) suggested a change from the traditional mainframe computer / FORTRAN environment. These ideas were incorporated into the AIRNET program (Walton, 1989). The primary purpose of AIRNET was to show that a quick, simple, and cheap solution existed for airflow network problems.

At the same time, work was being done by Magnus Herrlin (1992) as part of the COMIS (Conjunction of Multizone Infiltration Specialists) project. Magnus was using a band-solver for the linear equations and a method which determined an optimum under-relaxation coefficient, ω , to improve the convergence of the N-R solution. This method tended to need significantly fewer iterations than AIRNET's Steffensen method (using $\omega = 1.0$ or $\omega = 0.5$ depending on conditions) indicating a better, and probably safer, method. On the other hand, the time required to compute ω at each iteration was of the same order as one N-R iteration, and it required that all components use the same powerlaw equation.

The effectiveness of an optimum, variable ω suggested a little research into using a less time consuming constant ω . For example, Demuren (1986) used a constant ω of 0.5 to prevent oscillations in the N-R iterations. Several tests indicated an ω of about 0.75 was appropriate for a range of airflow problems and was much better than AIRNET's original Steffensen method. This idea was also tested by Wray (1991) in ASHRAE research, 618-RP, to determine the best algorithm for estimating pressures and airflows for the design of smoke control systems. In addition to the AIRNET algorithm, two loop based methods common to hydraulic analysis were tested. All three methods gave identical answers. AIRNET was fastest. This is not to say that the AIRNET method would be better for hydraulic problems. There are significant differences in the configuration of typical airflow versus hydraulic networks. Hydraulic networks tend to require significantly fewer simultaneous loop equation than simultaneous node equations. Airflow networks use similar numbers of loop or node equations. The skyline sparse matrix technique is ideally suited to tall buildings where the simple heuristic of grouping nodes in order by building floor automatically produces a factorable sparse Jacobian matrix.

A final comparison to Herrlin's optimized ω has not yet been done. CONTAM will use the constant ω method because of its speed and its flexibility in defining airflow elements. The current COMIS computer program contains several features such as thermal stratification (Feustel and Rayner-Hooson, 1990), some component models (Allard & Utsumi, 1992), and a wind pressure algorithm (Grosso, 1992), which could usefully be included in CONTAM94.

During the work on ASHRAE 618-RP another small change was made AIRNET's implementation of Newton's method. The test cases included an atrium which was modeled as a vertical group of zones connected by extremely large openings. Direct implementation of equation (25) as part of the calculation of element airflows was not sufficiently accurate for computing the terms in the Jacobian, even using double precision arithmetic. This was corrected by applying the correction term from equation (20) directly to ΔP for each flow path. This idea was originally suggested by Samuel Hassid (Technion, Israel) for use with the single precision calculation in TARP.

4. COMPUTING ENVIRONMENT

4.1 Requirements

CONTAM93 requires a 286 class (or higher) PC compatible computer with a math coprocessor, VGA graphics, and MS-DOS. The lowest level of MS-DOS usable has not been determined. CONTAM93 was developed on a machine running MS-DOS 5.0 and does not require special DOS procedures. CONTAM93 has been run as a DOS program under Windows 3.1. This combination of equipment and operating system was chosen because they are commonly available, adequate for the task at hand, and allowed quick development of the program. They limit the portability of the program.

CONTAM93 actually consists of two programs: CONTAMX is a non-interactive program which computes the airflows and contaminant dispersal. CONTAM is an interactive program for processing the CONTAMX input and output files. Both CONTAM and CONTAMX operate under the 640K-byte memory limit of MS-DOS, but this still leaves room for simulating some very large buildings.

4.2 Installation

The installation process is discussed in the README file on the distribution diskette. Be sure to read this file before installing or using the program. The README file will include the most current information on the installation and use of CONTAM93.

CONTAM93 can be installed on the hard disk in the following manner:

4.2.1 Create a subdirectory and then make it the default. For example, to install on the C: drive enter the following three commands:

```
C:                ( insures that you are on the C: drive )
MKDIR C:\CONTAM93 ( creates directory for CONTAM programs )
CHDIR C:\CONTAM93 ( shifts to CONTAM93 )
```

The directory containing the CONTAM93 programs will subsequently be referred to as CONTAM93, although you may use a different directory name if you wish.

4.2.2 Place the distribution diskette in drive A: or drive B: as appropriate. Then enter A:INSTALL A: (or B:INSTALL B:).

4.2.3 Answer the questions that the program asks (by pressing Y or N) to install the selected programs and/or data files onto your hard disk. The CONTAM.EXE, CONTAM.HLP, and CONTAMX.EXE files may be set as read-only files so they cannot be accidentally modified or erased. This is done with the DOS ATTRIB command.

4.2.4 The source code used to develop CONTAM93 is included on the distribution diskette. The source files (.CMB extension) require additional processing. Each should be copied to a separate directory and there separated into subfiles with the CMB.EXE program. Some of the programs need "include" files that are in CONTAM.CMB. The programs include MAKE files for compilation by Borland C++ (version 3.1). Some of the batch files refer to the directory structure used during program development:

```
C:\BC                all C related files stored under this level
C:\BC\CONTAM93       all CONTAM93 files consolidated here
C:\BC\CONTAM93\CONTAM  CONTAM.EXE created here
C:\BC\CONTAM93\CONTAM\HELP  CONTAM.HLP created here
C:\BC\CONTAM93\CONTAMX  CONTAMX.EXE created here
C:\BC\CONTAM93\CTOOLS   miscellaneous utilities compiled here
```

Replace batch file references to "see" with your own ASCII editor.

4.2.5 To remove CONTAM93 follow these steps:

CHDIR to the CONTAM93 directory.

To remove the read-only protection enter: ATTRIB -r *.*

Delete all files in CONTAM93 in the usual manner: DEL *.*

Delete project related files in other directories.

Fix the PATH command in your AUTOEXEC.BAT file, if you have added the path to CONTAM93 there.

Fix any changes to your CONFIG.SYS file.

4.3 CONTAM93 files

4.3.1 Files installed into the CONTAM93 directory:

```
CONTAM.EXE  The CONTAM93 input/output processor.
CONTAM.HLP  Data entry and help screens for CONTAM.EXE.
CONTAMX.EXE The CONTAM93 simulation program.
APPDIR.EXE  A program to temporarily modify the PATH.
README     The README file mentioned above.
```

4.3.2 Permanent file possibly added to the CONTAM93 directory:

```
CONTAM.CFG  User specified sensitivity of the keyboard and mouse.
```

4.3.3 Temporary files added to the CONTAM93 directory:

CONTAM.TMP	Current zone and path data.
CONTAM.WDW	Used to save and restore screen images.
CONTAM.LST	Information (e.g. error messages) from latest CONTAM run.
CONTAM.KEY	A copy of all keystrokes during latest CONTAM run.
CONTAMX.LST	Information (e.g. error messages) from latest CONTAMX run.

New temporary files overwrite the old files with every run. The last three files are useful for tracking down possible program or user errors.

4.3.4 Files that may be in a separate project directory:

<i>project</i> .PRJ	The project data file; <i>project</i> is the user-defined project name.
<i>project</i> .NDF	CONTAMX input file created by CONTAM.
<i>weather</i> .WTH	Weather file input to CONTAMX created by editing.
<i>project</i> .SIM	Simulation results file created by CONTAMX.
<i>project</i> .ZAP	Zone and path results report created by CONTAM.
<i>project</i> .CTM	Transient contaminant results report from CONTAM.
C.BAT	Automatically generated batch file used to start CONTAMX or CONTAM.

4.4 Running CONTAM93

4.4.1 The simplest way to run CONTAM occurs when the project files are in the CONTAM93 directory. CHDIR until CONTAM93 is the default directory and enter the word CONTAM to begin processing. Use the on-line help (press the F1 key) for information. Details on the input process are given in the Appendices.

4.4.2 If you plan to use an existing project file, a shortcut is to enter CONTAM *project*, where *project* is the project name (the name of the project file without the .PRJ extension).

4.4.3 CONTAM uses temporary files to allow it to handle very large projects within the DOS 640K byte memory limit. Access to the temporary files can be significantly improved by using the DOS SMARTDrive disk-caching program. Add a line to your CONFIG.SYS file similar to:

```
DEVICE=C:\DOS\SMARTDRV.SYS 2048 1024
```

Reserve at least 1 M-byte, if possible. See your MS-DOS manual for details.

4.4.4 If you wish to run CONTAM93 from a different directory, add the CONTAM93 directory to your path. This is useful if you are working on multiple projects and wish to keep data files related to each project in a separate directory. The path addition can be done permanently or only when you want to run CONTAM93 by running the APPDIR program (file APPDIR.EXE) whenever you enter the CONTAM93 directory. Then CHDIR to the directory containing the desired project files.

It may be desirable to put the above actions into a simple batch file which would reside in a directory in your default path. For example, create a file named C93.BAT consisting of the following three lines:

```
CD C:\CONTAM93 ( switch to CONTAM93 directory )
APPDIR          ( append directory to current path )
CD %1          ( switch to directory containing project files )
```

Then to reach a particular "project" directory, you would only have to enter:

```
C93 project ( where project is the user-defined project name )
```

4.4.5 If a problem occurs while running CONTAM, copy the CONTAM.LST and CONTAM.KEY files to a diskette and take it to a program consultant. If a problem occurs while running CONTAMX, copy the CONTAMX.LST, *project.PRJ*, and *project.NDF* files to a diskette. These files will allow a recreation of the problem.

4.4.6 It was originally intended that CONTAM and CONTAMX invoke each other automatically, but for some unknown reason this did not work consistently. A simple solution has been to let each program write a small batch file, C.BAT, which lets you start the other program with only two key strokes.

5. FUTURE DEVELOPMENTS

CONTAM93, as its name suggests, is an interim product which will be expanded to include new simulation capabilities and usability features.

Additional simulation capabilities might include:

- Perform exposure analysis. This would make the program more useful for air quality analysis.
- Allow a detailed description of the air handling system. Will use a sketch of the ductwork to compute system airflows.
- Allow contaminant concentrations to be a significant portion of the total mass in the zone. This would handle water vapor using the psychrometric equations. It might also be useful for a simple model of "smoke" defined as the total gaseous and particulate products of combustion.
- Model thermal mass. This might include a simple thermal model with heat transfer to and from the air nodes which could be useful with a simple fire model.
- Model thermal stratification.
- Include models from the COMIS program (Feustel & Rayner-Hooson, 1990) and the IEA ANNEX 23 work.

Additional usability features might include:

- Expand printer output to include graphic files similar to the screen graphs to document the input data.
- Add graphic and printer output detailing flows and pressures in stairwell and elevator shafts for use with smoke control analysis.
- Create the weather file from within CONTAM. Presently the file must be created with an ASCII editor. Creation from CONTAM would insure that the right contaminants (and their units) are being used.
- Develop a global library of airflow and contaminant elements. This would be a library of components available for any CONTAM run and would include the capability of moving an item from the library to the project and vice versa.
- Implement a parametric wind pressure model such as that used in COMIS (Grosso, 1992).

Input from program users is welcomed.

6. ACKNOWLEDGMENTS

The programs described in this report were developed under the funding of the National Institute of Standards and Technology. The author wishes to thank Andy Persily, John Klote, Jim Axley, Steve Emmerich, Jin Fang, and Cherie Bulala for their work in reviewing the programs and their suggestions for improvements.

7. REFERENCES

Allard, F. & Y. Utsumi. 1992. "Airflow through large openings", *Energy and Buildings*, Vol 18, pp 133-146.

ASHRAE. 1993. *ASHRAE Handbook - 1989 Fundamentals*, ASHRAE, Atlanta GA.

Axley, J. 1987. "Indoor Air Quality Modeling Phase II Report", NBSIR 87-3661, National Bureau of Standards (U.S.).

Axley, J. 1988. "Progress Toward a General Analytical Method for Predicting Indoor Air Pollution in Buildings, Indoor Air Quality Modeling Phase III Report", NBSIR 88-3814, National Bureau of Standards (U.S.).

Conte, S.D. and C. de Boor. 1972. *Elementary Numerical Analysis*, McGraw-Hill, New York NY, pp 299-306.

Cross, H. 1934. "Analysis of Flow in Networks of Conduits or Conductors", University of Illinois, Engineering Experiment Station, Bulletin #286.

Demuren, A.O. & F.J.K. Ideriah. 1986. "Pipe Network Analysis by Partial Pivoting Method", *Journal of Hydraulic Engineering*, ASCE, Vol. 112, pp 327-333.

Dhatt, G., G. Touzot, & G. Catin. 1984. *The Finite Element Method Displayed*, John Wiley & Sons, New York.

Feustel, H.E. & J. Dieris. 1992. "A survey of airflow models for multizone structures", *Energy and Buildings*, Vol 18, pp 79-100.

Feustel, H.E. & A. Rayner-Hooson (Eds). 1990. "COMIS Fundamentals", Lawrence Berkeley Laboratory, report LBL-28560.

Grosso, M. 1992. "Wind pressure distribution around buildings: a parametrical model", *Energy and Buildings*, Vol 18, pp 101-132.

Grot, R.A. 1991. "User Manual NBSAVIS CONTAM88", NISTIR 4585, National Institute of Standards and Technology.

Herrlin, M.K. 1992. "Solution methods for the air balance in multizone buildings", *Energy and Buildings*, Vol 18, pp 159-170.

Isaacs, L.T. & K.G. Mills. 1980. "Linear Theory Methods for Pipe Network Analysis", *Journal of the*

Hydraulics Division, Proceedings ASCE, Vol. 106, pp 1191-1201. (See also author's closure, 1982, Vol. 108, p 153)

Klote, J.H. 1981. "A Computer Program for Analysis of Smoke Control Systems", NBSIR 80-2157, National Bureau of Standards (U.S.).

Klote, J.H. & J.W. Fothergill Jr. 1983. *Design of Smoke Control Systems for Buildings*, ASHRAE, Atlanta GA.

Klote, J.H. & J.A. Milke. 1992. *Design of Smoke Management Systems*, ASHRAE, Atlanta GA.

Lam, C.F. & M.L. Wolla 1972. "Computer Analysis of Water Distribution systems: Part I -- Formulation of Equations", *Journal of the Hydraulics Division*, ASCE, Vol. 98, pp 335-344.

Martin, D.W. & G. Peters. 1963. "The Application of Newton's Method to Network Analysis by Digital Computer", *Journal of the Institute of Water Engineers*, Vol 17, pp 115-129.

Sander, D.M. 1974. "Fortran IV Program to Compute Air Infiltration in Buildings", NRC Canada, Division of Building Research, Computer Program No. 37.

Walton, G.N. 1983. "Thermal Analysis Research Program Reference Manual", NBSIR 83-2655, National Bureau of Standards (U.S.).

Walton, G.N. 1984. "A Computer Algorithm for Predicting Infiltration and Interroom Airflows," *ASHRAE Transactions*, Vol. 90, Part 1.

Walton, G.N. 1989. "AIRNET - A Computer Program for Building Airflow Network Modeling", NISTIR 89-4072, National Institute of Standards and Technology.

Wood, D.J. & A.G. Rayes. 1981. "Reliability of Algorithms for Pipe Network Analysis", *Journal of the Hydraulics Division*, ASCE, Vol. 107, pp 1145-1161.

Wray, C.P. & G.K. Yuill. 1993. "An Evaluation of Algorithms for Analyzing Smoke Control Systems", *ASHRAE Transactions*, Vol 99, Pt 1.

APPENDIX A: General Description

CONTAM93 consists of two programs: CONTAMX is a non-interactive program which computes the airflows and contaminant dispersal. CONTAM is an interactive program for processing the CONTAMX input and output files. Figure 1 indicates the relationships between the two programs and the various data files.

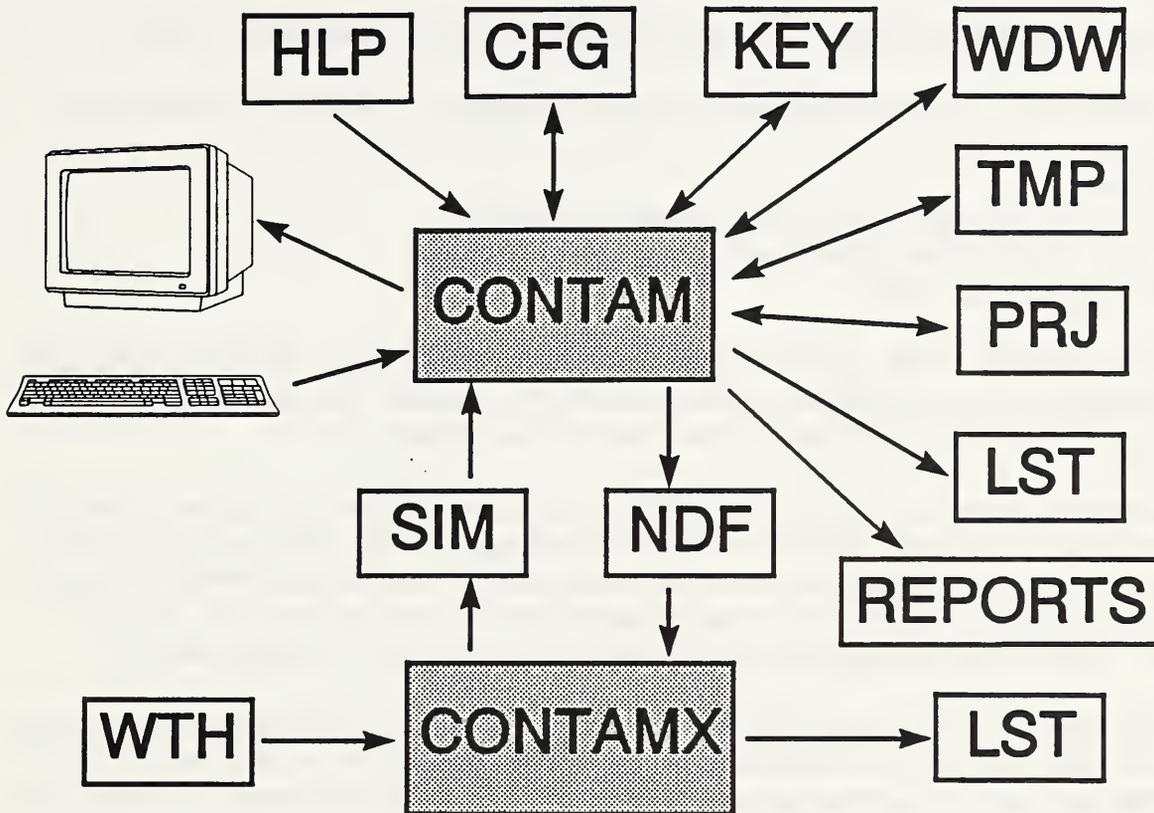


Figure 1. CONTAM programs and files

HLP CONTAM.HLP holds the data entry and help screens for CONTAM.EXE. This file must be present and may not be modified.

CFG CONTAM.CFG saves the user specified sensitivity of the keyboard and mouse.

KEY CONTAM.KEY saves a copy of all keystrokes during latest CONTAM run. This file can be used to recreate the user input to CONTAM for tracing program or user errors. It is a binary file.

WDW CONTAM.WDW is used to save and restore screen images.

TMP CONTAM.TMP holds the current zone and path data. This data is written to a disk file instead of RAM to allow large project descriptions within the 640K DOS limit. Its use is completely internal to CONTAM.

PRJ *project*.PRJ is the project data file; *project* is the user-defined project name. This file

contains the complete building description.

LST *CONTAM.LST* saves information (e.g. error messages) from latest CONTAM run.

NDF *project.NDF* is the CONTAMX input file created by CONTAM. It is smaller than the PRJ file because it contains only the data necessary for simulation.

SIM *project.SIM* is the simulation results file created by CONTAMX.

WTH *weather.WTH* is the weather file input to CONTAMX. It is created with an editor.

LST *CONTAMX.LST* saves information (e.g. error messages) from latest CONTAMX run.

REPORT files:

project.ZAP - Zone and path results report created by CONTAM.

project.CTM - Transient contaminant results report from CONTAM.

Spreadsheet program data files.

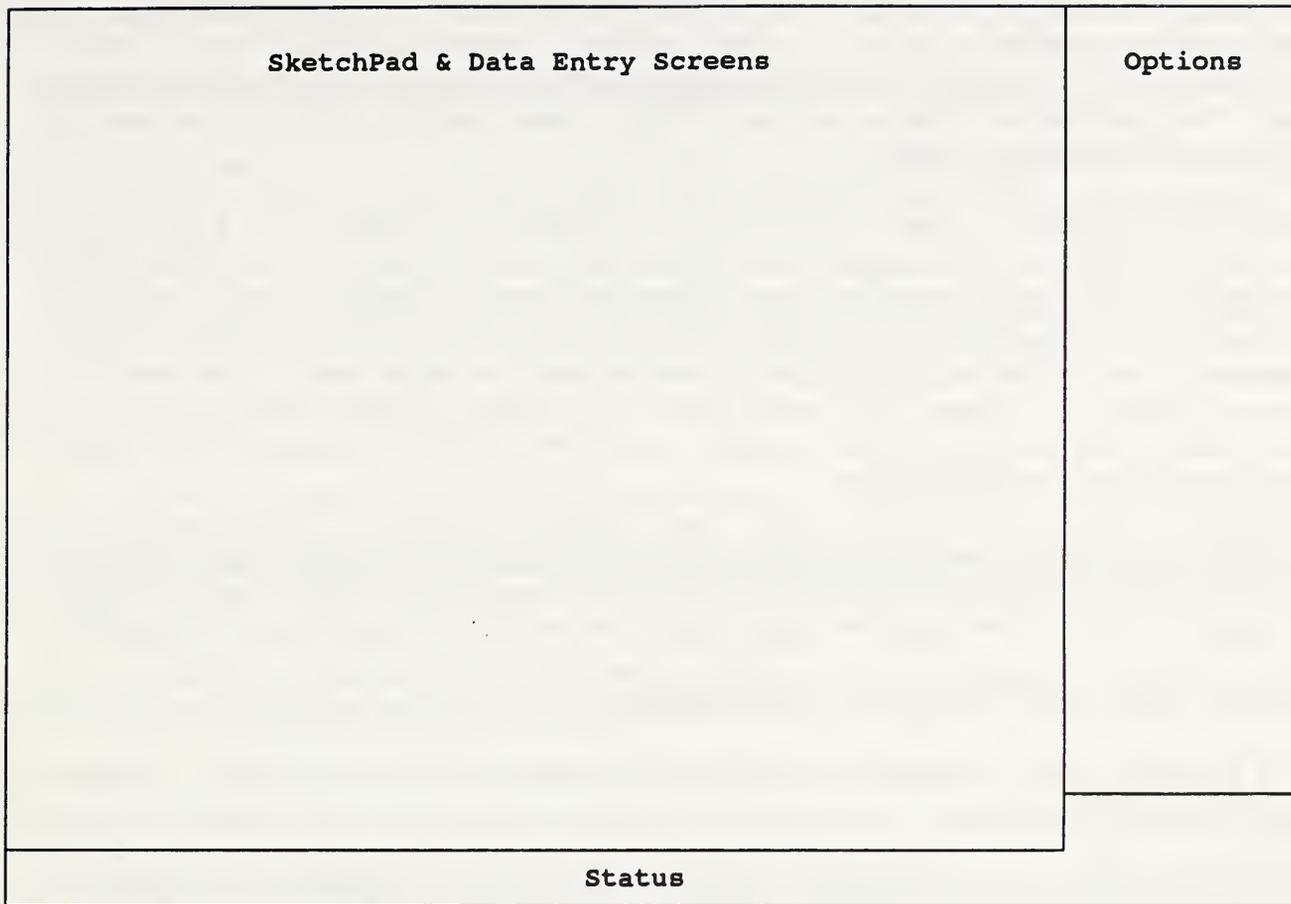
The HLP, KEY, WDW, TMP, and SIM files are recorded in binary mode. The WTH, CFG, PRJ, LST, and REPORT files are formatted and can be read with an editor. The only file which you should create or modify with an editor is the weather file (see Appendix D).

While CONTAM is running, the HLP, KEY, WDW, TMP, and LST files (in addition to the default console files) are always open. While CONTAMX is running, the WTH, SIM, and LST files are always open. A power interruption or reboot will cause whatever data has been written to those files to be lost as "lost allocation units" which can be cleared from the disk with the DOS CHKDSK command. Other files are opened only briefly to reduce the chances of a corrupted data file.

The KEY and two LST files can be useful to a program consultant in tracking down program or user errors. The WDW and TMP are completely internal to CONTAM and of no other use. The NDF and SIM files have no other use than to pass information between CONTAM and CONTAMX. The PRJ files must be saved for a record of what was simulated. The various reports provide only the results of simulation, not the input. A research or design project will usually involve several different project files describing different conditions from which comparisons and choices will be made.

APPENDIX B: The CONTAM User Interface

The screen is divided into three areas:



The main portion of the screen will be occupied by the SketchPad or a data entry screen as described on the following pages. The options area indicates possible actions that can be taken by pressing the keys indicated (in yellow). The status area displays information about the SketchPad such as the level name, error messages (which must be acknowledged by pressing any key), and questions (which must be acknowledged by pressing the Y or N key).

In the following discussion and in the options window some of the keys have special symbols:

↑ ↓ → ← cursor keys

Insert Delete Home End PgUp (page up) PgDn (page down) editing keys

↵ the enter (or return) key

Escape the "escape" key

^ the control key, used by pressing it and another key together, e.g. ^C

The mouse may be used in place of a few of these keys. The left button corresponds to the ↵ key; the right button to the Esc key. Vertical motion corresponds to the ↑ and ↓ cursor keys. On the SketchPad, horizontal motion corresponds to the → and ← cursor keys. On the data screens, horizontal motion corresponds to the ↑ and ↓ cursor keys. In general, the mouse is best used for reviewing SketchPad and data screen data, while the keyboard is best for creating or revising data.

THE SKETCHPAD

The description of the building is created (or modified) in the "SketchPad". The SketchPad is used to establish the geometric relationships of the relevant building features. It is not intended to produce a scale drawing of the building. It should be used to create a simplified model where the walls, zones, and air flow paths are topologically similar to the actual building. Engineering judgement must be used to define this model of the building. The SketchPad allows the entry and display of this data in an intuitive manner. The SketchPad will bring up various data entry screens needed to define the mathematical characteristics of the building features.

Building data is organized by "levels" with data entry beginning at the lowest level. A level would typically be a building floor, but a suspended ceiling acting as a return air plenum or a raised floor acting as a supply plenum may also be treated as a level leading to multiple levels per floor. Use the **PgUp** and **PgDn** keys to move between levels. The display includes features on the current level in dark colors, and features on the level below in light colors. When the levels are similar, most of the lower level features are hidden by the features on the current level. If you attempt to **PgUp** from the highest level, you may create a new level there. You may define a roof level which has no zones but does include openings from the level below to ambient.

As you move the cursor across zone and path symbols summary data is displayed on the status line. Pressing **←** while at a zone, path, or contaminant symbol gives access to the complete data and permits you to change the data. Other keys allow you to copy, move, or delete various (but not all) symbols.

It is probably best to sketch a level in the following order:

Press **W** to define the walls (envelope and partitions) which separate the zones on this level. The walls must be defined before the zones.

Press **Z** to define zones. A zone indicates a volume of air with uniform temperature and contaminant concentration. The zones must be defined before any of the remaining features.

Press **F** to define the air flow paths between zones. Path symbols placed on the walls are used to represent openings between zones or to ambient; any other placement represents an opening in the floor to the zone on the level below.

Press **A** to define a simple air handling system or supply or return points.

Press **S** to define contaminant sources or sinks.

Press **L** to redefine the level data; e.g. level elevation used in determining stack effect. This data is first set when you begin a new level.

When creating a new project data file, guard against data loss due to a power interruption by exiting this section (press **Esc**) and saving the project file (press **←**). You may then return to where you left the SketchPad (press **←**).

Walls — | ⌒ ⌓ ⌔ ⌕ ⌖ ⌗ ⌘ ⌙ ⌚

"Walls" are used to separate regions of uniform air on a given level. Such walls include the building envelope and internal partitions with a significant resistance to air flow. Walls must be drawn to create enclosed regions – the zones.

There is a set of implicit walls (actually floors) separating the zones on different levels.

Walls may be drawn in two ways. Press **W** to draw almost any shape wall using the cursor arrow keys. Such a wall cannot have an open or dangling end. Press **B** to draw a box. Follow the instructions in the options screen.

A wall can be deleted by pressing **Del**. The section of wall to be deleted is highlighted for confirmation.

Zones ☒ ☐ ☐

A "zone" indicates a volume of air with uniform temperature and contaminant concentration. Zones are defined on the SketchPad as a region surrounded by walls. Every zone must be identified. The ambient zone which surrounds the building has already been identified by the symbol at the lower-left corner of the sketch.

A zone is defined by moving the cursor to the desired position in the walled region, adding the desired zone symbol, and completing the zone data entry screen.

N, P, A:

Press **N** for a "normal" zone: separated by a floor from the zone below. Press **P** for a "phantom" zone which means there is no floor between this zone and the zone below. That is, the region on this level is actually part of the zone below. Press **A** for an "ambient" zone which is connected by some as yet undefined path to the implicit ambient zone. Such a feature is required to represent a courtyard. You may not place two zone symbols in the same region.

You may revise the zone data by pressing ← while the cursor is on the zone symbol. You may remove a zone symbol by pressing **Del**. You may move a zone symbol by pressing **M** and following the instructions in the options screen. Every zone must be defined before you can define airflow paths, system supply and return points, or contaminant sources/sinks.

Paths ◇

An airflow path indicates some building feature by which air can move from one zone to another. Such features include nearly invisible cracks in the building envelope, open doorways, and fans installed to exhaust contaminated air.

Path symbols placed on the walls are used to represent openings between zones or to ambient; any other placement represents an opening in the floor to the zone on the level below.

You may revise the path data by pressing ←. You may remove a path symbol by pressing Del. You may move a path symbol by pressing M and following the instructions in the options screen. You may create a path by copying a path symbol to a similar region on the sketch. Press C and follow the instructions in the options screen. Then review the path data entry screen.

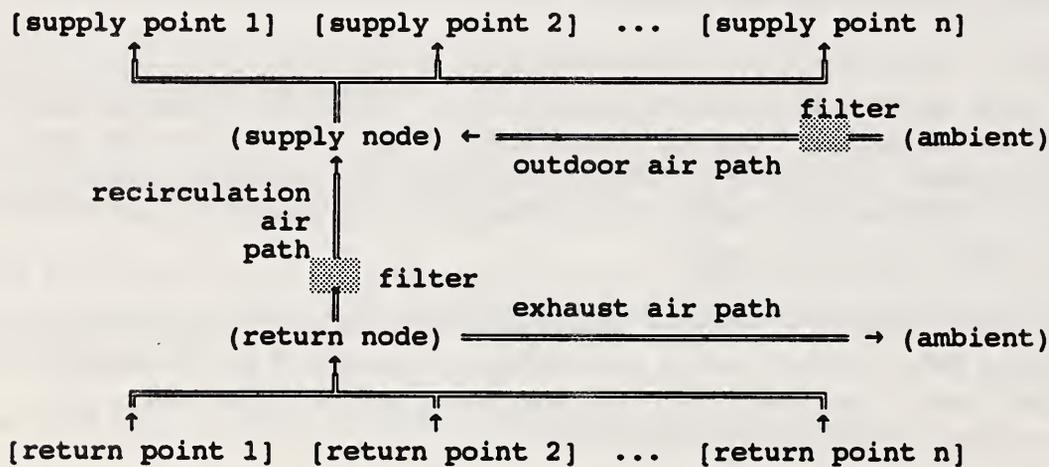
Contaminant □

Contaminant source (or sink) symbols may be placed in any zone. These represent any feature which produces or removes a contaminant within the list of available models.

You may revise the contaminant data by pressing ←. You may remove a contaminant symbol by pressing Del. You may move a contaminant symbol by pressing M and following the instructions in the options screen. You may create a contaminant by copying a contaminant symbol to a similar region on the sketch. Press C and follow the instructions in the options screen. Then review the contaminant data entry screen.

System S supply/return points □ □

The simple air handling system consists of 2 implicit airflow nodes (return and supply), 3 implicit flow paths (recirculation, outdoor, and exhaust air), and multiple supply and return points in the building zones:



The implicit nodes and paths are part of the AHS symbol which may be placed in any blank cell on the SketchPad. Supply and return point symbols must be placed within the appropriate zones. The system must be defined before any related supply/return points may be assigned.

DATA ENTRY SCREENS

In addition to the relationships established on the SketchPad, a great deal of numeric data is required to define the mathematical model for the simulation. This data is prepared in the data entry screens. A data entry screen consists of one or more data entry fields (designated by a special color: light gray on the blue background) and accompanying text. The data entry fields are generally organized with the most important data toward the top of the screen. A more detailed explanation of the data entries is available by pressing **F1**. If you are unsure about a value, leave the default. Reminders about how to use the data entry screen are provided in the options window to the right.

There are several different types of data entry fields which allow an action to be taken, a choice made, a name given, or a numeric value specified. Keyboard input is required from the user whenever the cursor is in a data entry field. The field may be blank or it may present a default response. The contents of a data entry field (even all blank) may be edited. That is, characters may be overwritten, deleted, or inserted. Press **←** to pass the data to the program. Program execution may be terminated when the cursor is in a data entry field by pressing **^C**.

There are usually several data entry fields on a single screen. These fields are ordered from top to bottom and left to right in a linear and circular manner. That is, the last field on the screen can be accessed as if it were above the first field and the first field as if it were below the last field. Movement between fields is done with the mouse or the **↑** and **↓** keys. Use the **PgUp** and **PgDn** keys to move directly to the first and last fields. The **Esc** key can also usually be used to reach the last field. It may or may not be necessary to make an entry in a field depending on context. When an entry is required but has not been made, you will be notified by an error message.

At the bottom of most data entry screens are the options "done" and "quit". "done" indicates the entries you have made will be saved; "quit" means they will not. If you entered this screen only to review the data, note that by pressing **Esc** (or the right mouse button) twice you will return to the prior screen without changing any data.

Data Entry Fields

To activate an action field (a one-character-wide field indicated by a check mark), press **←** (or the left mouse button) while at that field.

A choice of one among two or more options is indicated by several one-character-wide fields containing boxes , one with a dot inside indicating the current, or default, choice. You may (1) move through the choice fields, (2) accept the default by pressing **Esc**, or (3) move the cursor to the desired choice and press **←**.

Other data entry fields are longer with the size of the field indicating the maximum number of characters which may be entered. Data entry begins in the exchange mode, i.e. the value of the key pressed replaces the character at the cursor. Pressing **Ins** will switch to the insert mode (and from insert to exchange). Pressing **Del** will remove the character at the cursor. Pressing **^X** will clear the entire data entry field. Move the cursor to the right with the **→**, **Tab**, or **End** keys. Move the cursor left with the **←**, **shift-Tab**, or **Home** keys. When the edited data is satisfactory, press **←** which accepts the data and passes you to the next data field. **^←** accepts the data and passes you to the prior field. The use of any command causing a move between fields before pressing **←** (or **^←**) will restore the field to its original

value. The mouse may not be used to edit data in the field, but it may be used to exit the field in place of the \leftarrow , Esc, \uparrow , or \downarrow keys.

When numerical data is required, a proper number must be entered. In some cases the value must be an integer (e.g. 1000). In other cases a real value with a decimal point (1000.0) and/or scientific notation (1.0e3) is acceptable. If the characters you type are not a valid number, or if the number is physically impossible, an error message results. The error must be resolved before proceeding to another field.

Data entries representing physical values also have the units displayed in brackets, e.g. [m²]. Press F2 to change the units by following the directions in the options area. After the units have been selected, the original value in the data entry field will be converted and displayed in the new units.

If the desired input value is some arithmetic combination of known values, press F3 to activate an on-screen calculator which will enter the results of the calculation directly into the field.

In order to minimize the amount of numerical data entered, data representing a single building feature, such as a doorway, is grouped together and given a name by the user. When that feature is needed at another place in the building, it can be referred to by name instead of re-entering all the numeric data. Such a *name field* is indicated by <...> within it. Whether an entry is required or what no entry would mean is indicated within the <...>. For example, the following entries appear for defining an airflow path:

Flow element name: <req'd>

Filter name: <none>

These indicate that a flow element name is required but that a filter name is not. Failure to enter a filter name will simply mean there is no filter in this particular path.

When a named feature is required, and the feature has not been previously defined or you do not remember the name, press \leftarrow with the field clear (or with <...>) to define the feature. This will allow you to select a previously defined feature or create an entirely new one. Remember, you can clear the field by pressing ^X. Pressing \leftarrow with a name already present, displays the completed data entry screen for the feature.

APPENDIX C: CONTAM Data Entry Screens and Help Messages

The CONTAM data entry screens and associated help screens are reviewed on the following pages. Each screen is identified by a number, e.g. [040], in the upper-right corner. Actions that may cause transfer to another screen are shown by ---->. User specified units are shown by []. There tends to be a lot of redundancy in the help screens; each is designed to be fairly complete.

```

CONTAM - 93
General Information
Set Project Data          -----> [044]
Save Project File (project)
SketchPad Input
Component Library        not available
Run Simulation           -----> [120]
Analyze Results         -----> [140]
Set Program Parameters   -----> [042]
Quit
```

[040]

[041]

This is the main menu of CONTAM93 from which all data processing options can be reached.

GENERAL INFORMATION: Enter this section for an overview of the program. It will be especially useful the first time you run CONTAM93. The general information is presented by the help processor. Also note that the right side of the screen displays the options currently available to you. This tells you how to exit help.

PROJECT DATA: Processing normally starts in this section where you define the data files and set several default values.

SAVE PROJECT FILE: When creating a new project data file, you may "Save Project File" often to guard against data loss due to a power interruption.

SKETCHPAD INPUT: The description of the building is created, modified, or just reviewed in this section. It includes a combination of data entry screens for alpha-numeric data and a sketch to establish the connectivity in the airflow network. Note that when you exit from this section, you immediately have the option of saving the project file.

COMPONENT LIBRARY: This is a proposed extension for CONTAM-94. The component library is intended to allow access to a permanent library of airflow and contaminant elements, and editing of elements in the local library.

RUN SIMULATION: After completing the building description, enter this section to set up simulation control values, weather data, and initiate the simulation program, CONTAMX.

ANALYZE RESULTS: Upon completion of the simulation run, you are ready to analyze the results in this section.

PROGRAM PARAMETERS: The program parameters allow you to set the sensitivity of the keyboard and the mouse.

QUIT: Enter "Quit" to exit CONTAM. Note that quit does not save any files; you must explicitly save the project file.

[042]

Program Parameters

keyboard repeat delay:
keyboard repeat rate:

mouse motion for data entry:
mouse motion for SketchPad:

 right-handed mouse
 left-handed mouse

 done

[043]

The program parameters allow you to set the sensitivity of the keyboard and the mouse.

KEYBOARD: When you hold a key down, the keystroke is repeated to the screen. The repeating is controlled by two parameters: the delay before the first repeat is generated and the repeat rate.

Valid delay values are 0 (0.25 s) through 3 (1.00 s).

Valid repeat rates are 0 (30 char/s) through 31 (2 char/s).

This keyboard sensitivity persists after finishing CONTAM.

MOUSE: Mouse sensitivity is given in terms of the distance (measured in 10ths of a millimeter) the mouse must travel to move the cursor between data entry fields or between cells on the SketchPad. Smaller values give greater sensitivity.

Normally, the left button corresponds to the ← key and the right button to Escape. Specify a left-handed mouse to reverse that association.

DONE: These parameters are saved to a file named CONTAM.CFG so they can be retrieved the next time you run CONTAM. If this file is not found, the program reverts to the default settings.

[044]

Project Data

- █ Get or Define Project File -----> [046]
- █ Set Default Units and Values -----> [048]
- █ Define Contaminants -----> [050]
- █ Done

[045]

This menu gives access to some data which must be specified before any other data entry can take place. This data includes default and initial values, but the most important is the project file. Process the options on this menu in order.

PROJECT FILE: The building description is stored in a file: <project name>.PRJ You must keep track of the project names (= file names) to avoid over-writing files and losing data.

DEFAULTS: If you get an old project file, it is not necessary to set the default units, etc. All of that data will be taken from the old project file.

CONTAMINANTS: The contaminants and their default initial concentrations are defined.

[046]

Get or Define Project File

Old project name: █ (use existing name/file)

New project name: █ (create new name/file)

Description:

█ done

[047]

The building description is stored in a file: <project name>.PRJ You must keep track of the project names (= file names) to avoid over-writing files and losing data.

The project name may be up to 8 characters long, i.e. the length of a DOS file name. The ".PRJ" extension is added automatically by CONTAM. Project files are stored in the local (or working) directory. Because of the DOS file naming conventions, project names may not include : 's, \ 's, or blanks.

OLD PROJECT: You may retrieve previously prepared project data by entering an old project name, i.e. the name of an existing project file in the local directory. If you do not know the name, press ← with the field clear to search the list of project files in the local directory. You can clear the field by pressing ^X.

Once an old or new project has been named, the old project may not be renamed. (This would cause the old project file to be read thereby extending or over-writing existing data.)

NEW PROJECT: The "new project name" determines the file which will store the new data or changes you make in this work session. If the new project name is the same as the old project name, the data in the old project file will be over-written. You may or may not want that to happen.

DESCRIPTION: Use the project description to give a very brief summary of the contents of the project file, for example how it varies from the base case. This data will later help you search through old project files.

Project files can become rather large, but they are very compressible. A single design project will require many project files. When the design project is completed, archive the files to floppy disk using a file compression program, and remove the files from the local directory. This will help the project file search.

Set Default Units and Values

[048]

Default units: SI
 I-P

Default flows: [kg/s]
 [cfm]
 [L/s]
 [m³/s]
 [m³/h]
 [lb/s]
 [ft³/h]
 [L/min]

Default zone temperatures: []
Relative north: (deg)

done

[049]

UNITS: Most of the world, including NIST, uses S.I. units. However, in the United States it is still necessary to allow for other units. CONTAM permits a great deal of flexibility in units. Here you may specify the defaults. Most data entry screens allow you to make local changes in the units.

TEMPERATURE: Most building zones will probably be at the same temperature. Enter that value here. Changes for individual zones will be made later.

NORTH: The direction to relative north will be used relating wall angles on the SketchPad to true north for determining wind direction effects. Give the clockwise angle from vertical on the screen to true north.

[052]

Define Lowest Building Level

Name of level: []
Elevation of this level: []
Distance to next level: []
[] done

[053]

Building data is organized by "levels" with data entry beginning at the lowest level. A level would typically be a building floor, but a suspended ceiling acting as a return air plenum or a raised floor acting as a supply plenum may also be treated as a level leading to multiple levels per floor.

NAME: Each level must be given a unique (4 or fewer character) name. You will later be asked to refer to levels by name, so make the level name easy to remember and significant of the building level being described.

ELEVATION: Enter the elevation of this level relative to some building reference elevation. This reference elevation will be used in describing the elevations of all other levels -- it should probably be ground level. For consistency, let the elevation of this level refer to the elevation of the floor -- the lowest part of this level.

DISTANCE: Enter the vertical distance to the next level.

(elevation of this level) + (distance to next level)
= (elevation of next level)

This value will be used to set the default elevation of the next level.

[054]

Change Building Level Data

Name of level: []
Elevation of this level: []
Distance to next level: []
[] quit
[] done

[055]

This menu allows you to change the level data.

NAME: The level name may be replaced by another unique name.

ELEVATION/DISTANCE: If you change the elevation of this level (relative to the building reference elevation), then the distance above the level below must change. If you change the distance to the next level, then the elevations of all levels above this one must change. These changes will be done automatically.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not.

[056]

Define New Building Level

Name of level:

Copy from level: (optional)

Elevation of this level: []

Distance to next level: []

done

[057]

This data screen appears after you PgUp from the highest building level in order to create a new highest level.

NAME: Each level must be given a unique (4 or fewer character) name. You will later be asked to refer to levels by name, so make the level name easy to remember and significant of the building level being described.

COPY LEVEL: Since levels in tall buildings tend to be similar, you may copy the wall, zone, and path descriptions from any previously defined level to this level. Then you may edit the SketchPad to make any necessary adjustments in the description of this new level. It is very useful in reviewing the simulation results to keep the zone and path symbols on different levels at the same positions on the SketchPad.

ELEVATION: Enter the elevation of this level relative to the original building reference elevation -- probably the ground level. For consistency, let the elevation of this level refer to the elevation of the floor -- the lowest part of this level. The default shown is derived from data on the level below.

DISTANCE: Enter the vertical distance to the next level.
(elevation of this level) + (distance to next level)
= (elevation of next level)
This value sets the default elevation of the next level.

Zone Data

Name: (Level: name)

Temperature: []

Pressure: [] variable or constant

Volume: [] or floor area: []

Contaminant concentrations: variable or constant

Initial contaminant concentrations:

[]
 []
 []
 []
 []
 []
 []
 []
 []
 []

Kinetic reaction: -----> [064] | [066]

quit
 done

[061]

A zone indicates a volume of air with uniform temperature and contaminant concentration.

NAME: The zone name is displayed in the status window and on reports as a reminder to you of what the zone represents. Specify a name accordingly.

TEMPERATURE: The zone temperature will be used in determining stack effects.

PRESSURE: The zone air pressure relative to ambient can be specified as a constant. This option will probably never be needed in a real building, but it has been useful in the development of analytic test cases.

VOLUME: Zone volume is used in the dynamic contaminant calculations. Instead of the volume, you may enter a floor area which is multiplied by the height of the current building level to get a volume.

CONCENTRATIONS: Enter the initial contaminant concentrations for dynamic simulations. The defaults are set in the project data.

REACTIONS: If you know it, you may enter the name of a previously defined kinetic reaction. Otherwise, press \leftarrow with the field clear to find or define a reaction. Leaving this field blank indicates no reactions in the zone. Pressing \leftarrow with a kinetic reaction name already present, displays the completed data entry screen for that reaction.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not. If you entered this screen only to review the zone data, note that by pressing **Esc** twice you will return without changing any zone data.

Kinetic Reaction

- █ search list of existing reactions
- █ define a new reaction matrix -----> [066]
- █ quit

REACTION DATA: If you think that the reaction has already been defined, select "search list of existing reactions". This will display all reactions allowing you to select one. If it has not been defined, select "define a new reaction matrix" which will display a data entry screen for the creation of a new reaction.

QUIT: "quit" to return without defining a reaction.

Kinetic reaction name: █ (required)

Description: █

0 1 2 3 4

0 █

1

2

. . .

8

9

5 6 7 8 9

0

1

2

. . .

8

9

- █ done
- █ quit

This is a description of linear chemical reactions among the contaminants. The production or destruction of a contaminant is given by:

$$R(a) = \text{Sum } K(a,b) C(b)$$

where $K(a,b)$ are the reaction rate coefficients. CONTAM93 uses the convention that positive reaction coefficients produce an increase in contaminant concentration. The units of concentration are kg contaminant / kg air.

NAME: A unique name is required for every kinetic reaction for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this kinetic reaction later.

REACTION COEFFICIENTS: The reaction coefficients are displayed in matrix form with the matrix displayed in two section to get it on the screen.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a kinetic reaction has not been defined.

Airflow Path

```

Flow element name: <req'd> -----> [072] | [074]
Relative elevation: [ ]
Control Profile: <none> -----> [080] | [082]
Multiplier: [ ]
Filter name: <none> -----> [076] | [078]
Positive flow direction: [ ] from zone to zone
                        [ ] from zone to zone
Wind pressure: [ ] none -- [ ] constant -- [ ] variable -----> [090]
                                                | [092]
Limits: [ ] none -- [ ] pressure -- [ ] flow
minimum: [ ] [ ]
maximum: [ ] [ ]
[ ] quit
[ ] done

```

[071]

An airflow path indicates some building feature by which air can move from one zone to another. Such features include nearly invisible cracks in the building envelope, open doorways, and fans installed to exhaust contaminated air. The description of a flow path is the most complicated data entry in CONTAM.

The flow through most of these features is determined by the air pressures in the adjacent zones and the flow characteristics of the feature itself. In most buildings certain airflow features are repeated many times. Therefore, we treat the flow characteristics of a building feature separate from the path and call this a "flow element" which can be defined once and then used many times by name.

ELEMENT NAME: If you know it, you may enter the name of a previously defined flow element. Otherwise, press ← with the field clear to find or define a flow element. Remember, you can clear the field by pressing ^X. Pressing ← with a name already present, displays the completed data entry screen for that flow element.

ELEVATION: The elevation of the path will determine how it responds to, and influences, the building stack effect. Enter the elevation of the mid-height of the flow path minus the elevation of the current building level. The relative elevation for a path in the floor is typically zero.

CONTROL: The control profile is a special multiplier which can change the flow characteristics of the path as a function of time. How this is done depends on the flow element. If you know it, you may enter the name of a previously defined profile. Otherwise, press ← with the field clear to find or define a profile. Leaving this field blank indicates a control value of 1.0 at all times. Pressing ← with a profile name already present, displays the completed data entry screen for that control profile.

MULTIPLIER: The multiplier is a constant value allowing you, for example, to use a flow element description of one window and a multiplier of 5 to describe 5 windows in a particular wall.

If you have specified a "leakage area", complete the description here by giving the reference area, length, or number of units.

FILTER: If you know it, you may enter the name of a previously defined filter element. Otherwise, press ← with the field clear to find or define a filter element. Leaving this field blank indicates no filter in the path. Pressing ← with a filter name already present, displays the completed data entry screen for that filter.

POSITIVE DIRECTION: In the cases of fans, wind, and pressure/flow limits (below) it is necessary to know which direction is defined as positive. The arrows indicate the positive direction for pressure drop and flow. For a positive wind pressure to tend to force inward flow, the positive direction must be from ambient into the building.

WIND: There are three wind pressure options for building envelope surfaces: (1) no wind pressure, (2) a constant pressure, and (3) pressure dependent on wind speed and direction. Options (2) or (3) transfer you to the appropriate data entry screens.

LIMITS: In the design of smoke control systems there are often limits on the maximum and minimum pressure or flow on a path. For example, a stairwell door can require a minimum pressure drop from the stairwell to the zone to prevent smoke entering the evacuation route, but there is also a maximum pressure drop above which some people may not be able to open the door. You may specify either type of limits (default: no limits) and set the values. When you display the results of a simulation, those paths with pressures or flows outside the limits will be highlighted.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not. If you entered this screen only to review the zone data, note that by pressing Esc twice you will return without changing any zone data.

[072]

Airflow Elements

█ search list of existing elements

█ define a new element

-----> [074]

█ quit

[073]

ELEMENT DATA: If you think that the element has already been defined, select "search list of existing elements". This will display all elements allowing you to select one. If it has not been defined, select "define a new element" which will display a data entry screen for the creation of a new element.

QUIT: "quit" to return without defining an element.

Airflow Elements

One-way flow using powerlaw model:

leakage area data	connection (ASCOS) data	----->	[154]	/	[152]
orifice area data	crack description	----->	[150]	/	[180]
test data (1 point)	test data (2 points)	----->	[160]	/	[162]
stairwell		----->	[156]		

One-way flow using quadratic model:

crack description	test data (2 points)	----->	[166]	/	[164]
shaft		----->	[158]		

Two-way flow models:

one-opening	two-opening	----->	[168]	/	[170]
-------------	-------------	--------	-------	---	-------

Fan models:

constant mass flow	constant volume flow	----->	[172]	/	[174]
performance curve		----->	[176]		

quit

The powerlaw model permits airflow in only one direction through the path based on the direction of the pressure drop.

Powerlaw: $w = C (\Delta P)^n$ ΔP = pressure drop, w = mass flow rate

LEAKAGE AREA refers to a description of airflow features given in Chapter 23 of the 1993 ASHRAE Handbook of Fundamentals.

CONNECTION refers to the airflow description used in the ASCOS program.

ORIFICE AREA relates the opening description to the orifice equation.

CRACK describes a narrow opening by its length and width.

TEST (1) uses a single measured (ΔP , w) data point.

TEST (2) uses two measured (ΔP , w) data points.

STAIRWELL describes the resistance to vertical flow in a stairwell.

The quadratic model permits airflow in only one direction through the path based on the direction of the pressure drop.

Quadratic: $\Delta P = a w + b w^2$ ΔP = pressure drop, w = mass flow rate

CRACK describes a narrow opening in greater detail than above.

TEST (2) uses two measured (ΔP , w) data points.

SHAFT describes the resistance to flow in large (e.g. elevator) shafts.

The two-way flow models may be used to represent tall openings where the difference in the stack effect across the height of the opening might cause two-way flows.

The fan models should be used where known or mechanically forced flows are present.

QUIT: "quit" to return without defining an airflow element.

[080]

Control Profile

- █ search list of existing profiles
- █ define a new profile -----> [082]
- █ quit

See screen [073] for associated help.

[082]

Control Profile

Profile name: █ (required)

Description: █

- █ profile data -----> [084]
- █ quit
- █ done

[083]

Control profiles are used to modify various simulation features as a function of time. The control profile is displayed graphically at the bottom of that screen for a quick visual review of the data.

NAME: A unique name is required for every control profile for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this control profile later.

DATA: Enter this option to create or modify the values which define the control profile.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a control profile has not been defined.

[084]

time	value	time	value
█:█:█	█	█:█:█	█
█:█:█	█	█:█:█	█
...
█:█:█	█	█:█:█	█

█ done

[085]

You may enter up to 50 time/control value data points. Values are in the range 0 to 1. The first value must be at time 00:00:00. An identical value at time 24:00:00 is automatically assumed.

When you finish entering data, the data points will be sorted by time. That is, you do not have to enter the points in order. An old data point can be deleted by setting the time to 24:00:00.

[090]

Constant Wind Pressure

Wind pressure: []

quit
done

[091]

Enter a constant (wind speed and direction independent) wind pressure such as might be obtained from a single wind tunnel test.

[092]

Variable Wind Pressure

Wind pressure modifier: []

Wall azimuth angle: [] (deg)

Wind pressure profile: <req'd> -----> [096]

quit
done

[093]

Variable wind pressures are computed from:
(the dynamic pressure of the wind at some reference height) *
(a pressure coefficient accounting building/terrain effects) *
(a pressure coefficient accounting for relative wind direction)

MODIFIER: Enter a value to account for the building/terrain effects.

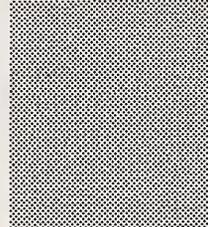
AZIMUTH: Enter the direction the wall faces (clockwise from north).

PROFILE: The wind pressure profile accounts for the wind direction effects. If you know it, you may enter the name of a previously defined profile. Otherwise, press ← with the field clear to find or define a profile. Pressing ← with a profile name already present, displays the completed data entry screen for that wind pressure profile.

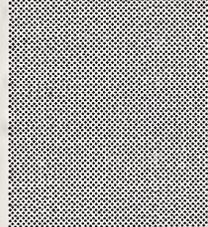
Wind Pressure Profile Name: (required)

Description:

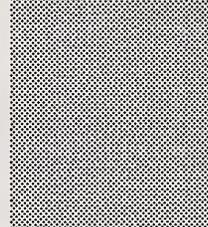
Angle (deg)



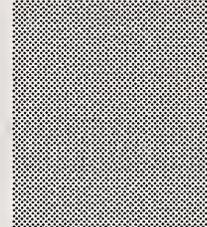
Coefficient



Angle (deg)



Coefficient



quit done graph

Wind pressure profiles are used to account for the wind direction effects. The profile is displayed graphically at the bottom of that screen for a quick visual review of the data.

NAME: A unique name is required for every wind pressure profile for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this profile later.

DATA: You may enter up to 16 azimuth/pressure coefficient points. Values are in the range 0 to 1. The first value must be at azimuth 0. An identical value at azimuth 360 is automatically assumed.

When you finish entering data, the data points will be sorted by angle. That is, you do not have to enter the points in order. An old data point can be deleted by setting the azimuth to 360.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a control profile has not been defined.

GRAPH: Graph the data before entering "done".

Wind Pressure Profile

- search list of existing profiles
- define a new profile -----> [094]
- quit

PROFILE DATA: If you think that the profile has already been defined, select "search list of existing profiles". This will display all profiles allowing you to select one. If it has not been defined, select "define a new profile" which will display a data entry screen for the creation of a new profile.

QUIT: "quit" to return without defining an profile.

[100]

Contaminant Source/Sink

Source/sink name: <req'd> -----> [102]

Control profile: <none> -----> [082]

Multiplier: []

[] quit
[] done

[101]

NAME: If you know it, you may enter the name of a previously defined source/sink model. Otherwise, press ← with the field clear to find or define a source/sink model. Remember, you can clear the field by pressing ^X. Pressing ← with a name already present, displays the completed data entry screen for that model.

CONTROL: The control profile is a special multiplier which can change the source/sink rate as a function of time. If you know it, you may enter the name of a previously defined profile. Otherwise, press ← with the field clear to find or define a profile. Leaving this field blank indicates a control value of 1.0 at all times. Pressing ← with a profile name already present, displays the completed data entry screen for that control profile.

MULTIPLIER: The multiplier is a constant value allowing you, for example, to use a model of an area source for 1 m² and a multiplier of 20 to describe an area source of 20 m².

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a contaminant source/sink has not been defined.

[102]

Contaminant Source/Sink Model

[] search list of existing models

[] define a new model -----> [104]

[] quit

MODEL DATA: If you think that the model has already been defined, select "search list of existing models". This will display all models allowing you to select one. If it has not been defined, select "define a new model" which will display a data entry screen for the creation of a new model.

QUIT: "quit" to return without defining a model.

Source/Sink Models

- █ create a simple coefficient model -----> [190]
- █ create a pressure driven model -----> [192]
- █ create a cutoff concentration model -----> [194]
- █ create a decaying source model -----> [196]
- █ create a boundary layer source/sink model -----> [198]
- █ quit

Select the desired model.

The general source/sink model allows constant contaminant generation and deposition rates.

The pressure driven model is for contaminant sources which are governed by the inside-outside pressure difference.

The cutoff concentration model may be appropriate for some volatile organic compound sources.

The decaying source model may be appropriate for some other volatile organic compound sources.

The boundary layer diffusion controlled source/sink model may be used to model a reversible source/sink.

QUIT: "quit" to return without defining a source/sink model.

Air Handling System

System name: (required)Description:

Outside air control profile: <reg'd> -----> [082]

Minimum outside air flow: []

Outdoor air filter name: <none> -----> [076] | [078]

Recirculation filter name: <none> -----> [076] | [078]

supply system contaminant data -----> [062]

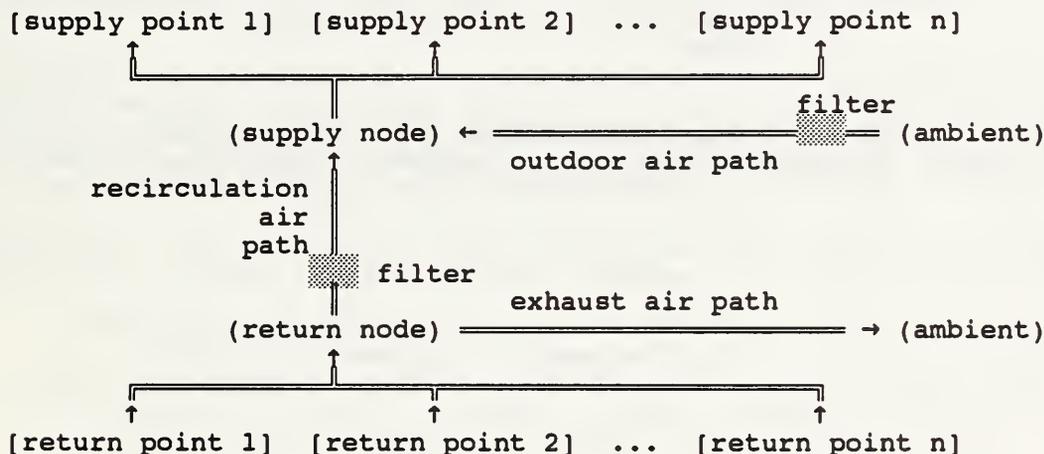
return system contaminant data -----> [062]

quit

done

[111]

The simple air handling system consists of 2 implicit airflow nodes (return and supply), 3 implicit flow paths (recirculation, outdoor, and exhaust air), and multiple supply and return points in the building zones:



NAME: A unique name is required for each system so it can be referenced later. The name should be descriptive, but since it is so short you may also enter a longer description to help you search for this system later.

OUTSIDE AIR: The system algorithm operates by summing all the supply flows, $\sum W_s$, and all the return flows, $\sum W_r$. The required amount of outside air is: $W_{oa}' = \max(F_{oa} * \sum W_s, \min(\sum W_s, M_{oa}))$, where

F_{oa} = the "fraction outside air" input parameter, and

M_{oa} = the "minimum outside air flow" input parameter.

The amount of recirculated air is: $W_{ra} = \min(\sum W_r, \sum W_s - W_{oa}')$.

The actual outside air is: $W_{oa} = \sum W_s - W_{ra}$.

The amount of exhaust air is: $W_{sa} = \sum W_r - W_{ra}$.

It is not necessary to have both supply and return points in a system. That is, you may create a system that only supplies air into or exhausts air from the building.

The minimum outside air flow is determined from the value given times the control value at the current time. If you know it, you may enter the name of

a previously defined profile. Otherwise, press ← with the field clear to find or define a profile. Leaving this field blank indicates a control value of 1.0 at all times. Pressing ← with a profile name already present, displays the completed data entry screen for that control profile.

FILTERS: A filter can be placed in the outside air path or the recirculation air path. If you know it, you may enter the name of a previously defined filter element. Otherwise, press ← with the field clear to find or define a filter element. Leaving this field blank indicates no filter in the path. Pressing ← with a filter name already present, displays the completed data entry screen for that filter.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an system has not been defined.

[112]

Supply or Return Point

```
System name:  <req'd>
Design flow rate:  [ ]
Control profile:  <none>      -----> [080] | [082]
Filter name:  <none>      -----> [076] | [078]
quit
done
```

[113]

SYSTEM NAME: The supply/return point must refer to a previously defined system. If you remember it, enter the name of the system. Otherwise, press ← with the field clear to search the list of system names. You can clear the field with ^X.

DESIGN FLOW: Enter the design, i.e. maximum, airflow rate.

CONTROL: The actual flow rate equals the design flow rate times the control value. If you know it, you may enter the name of a previously defined profile. Otherwise, press ← with the field clear to find or define a profile. Leaving this field blank indicates a control value of 1.0 at all times. Pressing ← with a name already present displays the completed data entry screen for that control profile.

FILTER: If you know it, you may enter the name of a previously defined filter element. Otherwise, press ← with the field clear to find or define a filter element. Leaving this field blank indicates no filter in the path. Pressing ← with a filter name already present, displays the completed data entry screen for that filter.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a supply/return point has not been defined.

System Contaminant Data

```

System volume: [ ]
Initial contaminant concentrations:
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]
[ ]
Kinetic reaction: [ ] -----> [064] | [066]
quit
done

```

VOLUME: System volume is used in the dynamic contaminant calculations.

CONCENTRATIONS: Enter the initial contaminant concentrations for dynamic simulations. The defaults are set in the project data.

REACTIONS: If you know it, you may enter the name of a previously defined kinetic reaction. Otherwise, press ← with the field clear to find or define a reaction. Leaving this field blank indicates no reactions in the zone. Pressing ← with a kinetic reaction name already present, displays the completed data entry screen for that reaction.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not.

Run Simulation

```

run control data          -----> [122]
weather data (all steady) -----> [124]
weather file (transient) -----> [126]
check wind pressures     -----> [128]
save project file
quit
done (begin simulation)

```

RUN CONTROL: In this section you define the type of simulation to be performed and several parameters that control the computational process.

WEATHER DATA: "Weather data" lets you set weather related variables such as temperature, wind speed, etc.

WEATHER FILE: This lets you specify the name of the weather file which is needed for a transient simulation.

WIND PRESSURES: This option allows you to review and revise the wind pressure data for the building envelope openings by using the SketchPad.

SAVE PROJECT FILE: If you save the project file here, it will preserve any changes made in the run control or weather data.

DONE/QUIT: "quit" will cause a return to the main menu. "done" will leave you ready to execute the simulation program, CONTAMX, by pressing C and ↵. When the simulation is done, you may resume CONTAM to analyze the results of the simulation.

Note: I have had some trouble with this part of the process. It appears that another program I use (which shall remain unnamed) messes up the RAM so that one program may not initiate the other one using function execl(). So I am letting each program write a small batch file, C.BAT, which starts the other with only two key strokes.

Run Control Data

Type of Simulation:

Air flows: steady or transient/cyclic
 Contaminants: none, steady, transient, or cyclic

Calculation Parameters:

maximum flow iterations:
 relaxation coefficient:
 relative flow convergence:
 absolute flow convergence: []
 time-of-day: : : (steady flow)
 initial time-of-day: : : (transient)
 final time-of-day: : : "
 relative C.C. convergence: (cyclic)
 absolute C.C. convergence: []
 simulation time step: [s]
 listing time step: [s]
 output dumps:

done

[123]

TYPE OF SIMULATION: You may specify either a steady or dynamic (transient / cyclic) calculation of the airflows. This is related to the contaminant simulation. A steady contaminant simulation can only use steady airflows. A transient simulation may run for up to 24 hours. A cyclic simulation repeats a 24-hour cycle until steady-periodic conditions are achieved. The latter two cases may use steady airflows with its consequent savings in computing time, but you are more likely to want dynamic airflows.

PARAMETERS:

"Maximum flow iterations" is used to prevent endless calculation if the flow iterations are not converging. This should be a problem only if you are using a bad relaxation coefficient.

The "relaxation coefficient" modifies the adjustment made by the pure Newton-Raphson method. Tests have found values near 0.75 to work well. Values near 1.0 sometimes lead to nonconvergent solutions. Values nearer 0.50 may be more reliable, but they take longer.

The flow iterations are determined to have converged when the sum of all flows into/out of the zone are less than the "relative flow convergence" times the sum of the absolute values of those flows. If the sum of the absolute values of the flows is less than "absolute flow convergence", the zone is assumed to be converged. This prevents a division by zero.

"Temperature adjustment" modifies the element flow coefficients to account for the actual properties of the air flowing through the elements. This may change the computed flows by a few percent. Its use in somewhat debatable since CONTAMX cannot account for any change in air temperature as it flows through the element.

A steady flow simulation uses the "time-of-day" to select control values for any controlled elements.

A transient flow/contaminant simulation begins at the "initial time-of-day" and ends at the "final time-of-day".

The cyclic simulation is determined to have converged when the relative

difference in the peak contaminant concentrations (C.C.) for all contaminants in all zones is less than the "relative C.C. convergence" value. "Absolute C.C. convergence" is used to prevent a division by zero as is done for flows.

The "simulation time step" is the constant time step for the fully implicit dynamic simulation. Shorter time steps are more accurate but also take more computation time.

CONTAMX creates output at every "listing time step" which must be some even multiple of the simulation time step. Again, shorter steps allow you to see the results in more detail but they require more computer time.

The "output dumps" value is a debugging tool where larger values cause more data to be written to the CONTAMX.LST file. You should leave this at 0.

[124]

Weather Data
Steady State Simulation

ambient temperature: []
absolute pressure: []
(or) site altitude: []
wind speed: []
wind direction: [] (deg)
done

[125]

TEMPERATURE: The ambient temperature will be used in evaluating outside air density and building stack effect.

PRESSURE / ALTITUDE: Be sure to use absolute barometric pressure instead of pressure corrected to sea level. If you do not know the absolute pressure, then enter the altitude of the building site, and a pressure will be computed.

WIND: Wind direction is the angle from which the wind blows measure clockwise from true north.

[126]

Weather File
Transient/Cyclic Simulation

Weather file name: <reg'd>

quit
done

[127]

The dynamic simulation require a weather file which contains time-dependent information of ambient temperature, wind speed and direction, and ambient contaminant concentrations.

The weather file must exist in the local directory. It is prepared external to CONTAM93. Enter the name of an existing weather file. If you do not know the name, press ← with the field clear to search the list of weather files in the local directory. You can clear the field by pressing ^X.

DONE/QUIT: Enter "done" to accept the weather file name. Enter "quit" to return to screen [120] without a weather file defined.

[128]

Wind Pressure Data

Positive flow direction: from zone to zone
from zone to zone

Wind pressure: none -- constant -- variable -----> [090]
| [092]
...

quit
done

[129]

Press enter while the cursor is at a path symbol on the envelope to review and revise the wind pressure data for that path. The magnitude and direction of the wind pressure are shown by the brown lines. The length of each line is proportional to the "scale" line shown at the bottom of the SketchPad. These wind pressures are based on current values of the steady-state wind speed and direction.

POSITIVE DIRECTION: In the cases of fans, wind, and pressure/flow limits (below) it is necessary to know which direction is defined as positive. The arrows indicate the positive direction for pressure drop and flow. For a positive wind pressure to tend to force inward flow, the positive direction must be from ambient into the building.

WIND: There are three wind pressure options for building envelope surfaces: (1) no wind pressure, (2) a constant pressure, and (3) pressure dependent on wind speed and direction. Options (2) or (3) transfer you to the appropriate data entry screens.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not.

Results

- ▣ display results on SketchPad
- ▣ write zone & path data output file (<project>.ZAP)
- ▣ write contaminant data output file (<project>.CTM)
- ▣ done

[141]

There are presently three options for processing the results of a simulation. You may choose an interactive display using the SketchPad or you may create files which can be edited or printed.

The zone and path file uses a format similar to the one used by the ASCOS program. It reports data only at a single time -- the time displayed when exiting the SketchPad. The default time is the start of the simulation. The file name is <project name>.ZAP

The contaminant data is printed in two different formats based on whether a steady or transient simulation was done. The transient file can be long. The file name is <project name>.CTM

(Proposed addition) An interactive display showing airflows in a column of zones (such as a stairwell).

Help for SketchPad display.

[143]

The SketchPad is used to display the results of the airflow simulation. Both the magnitude and direction of the computed airflows (blue lines) and pressure drops (brown lines) are shown. The length of each line is proportional to the "scale" lines shown at the bottom of the SketchPad. Paths where limits have been exceeded are highlighted in red.

Use the PgUp and PgDn keys to view the results on different levels. The level below the current level is normally show in gray. Press ^PgDn to turn on and off the display of this level.

When you move the cursor to a flow path symbol, the pressure drop and flow rate are displayed in the status window. For example, if you wish to check the flows through a stairwell door on all levels and the symbol for each door is at the same place on the SketchPad (which you get by copying levels), just move the cursor to the door path symbol, repeatedly press PgUp or PgDn, and review the data presented in the status window.

Use the End and Home keys to show the airflow data at later or earlier times. Note the time value above the level name at the bottom of the options window.

The zone contaminant concentrations can be displayed by moving the cursor to a zone symbol and pressing ←. For transient simulations, concentrations each normalized for the peak value are presented graphically. The transient pressure and flows for a path are displayed in a similar manner.

You can create a spreadsheet input file of the transient contaminant data for a single zone by pressing S and following the subsequent directions.

Help for transient contaminant data display

[144]

This screen displays the normalized contaminant concentrations in the zone as a function of time. Concentrations are normalized to the peak value of each contaminant during the simulation. This method was chosen instead of using the same scale for all contaminants because of the possibility of greatly differing magnitudes.

The contaminant names, peak concentrations, and concentration units are displayed above the graph. The selected contaminant is graphed in yellow instead of white.

Help for transient flows display

[145]

This screen displays the normalized flow rate(s) and pressure drop for the path as a function of time. Flows (yellow) and pressure drop (brown) are normalized to their peak values. This graph does not show much of interest if steady state flows were used in the simulation in which case the pressure curve is drawn over the flow curve.

Spreadsheet Data File for One Zone

[146]

File name:

[147]

The results of the transient contaminant simulation for a single zone will be written to a file in a format suitable for reading by a spreadsheet program. Header information is in double quotes and individual data elements are separated by commas.

Enter a file name which will allow you to access this data later. Note that the file does not contain zone identification.

If you cannot open a file (possibly due to a full disk) blank the file name field and quit.

Orifice Area → Powerlaw Model

Airflow element name: [] (required)

Description: []

Cross sectional area: [] []

Flow exponent: []

Discharge coefficient: []

Hydraulic diameter: [] []

Transition Reynolds number: []

use symbol for: [] small opening
[] large opening[] quit
[] done

Data describing an orifice is reduced to the powerlaw model:

$$w = C (\Delta P)^n \quad w = \text{mass flow rate, } \Delta P = \text{pressure drop}$$

This is the most flexible input for the powerlaw model, but other input forms use data more closely related to observable building features.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

AREA: Cross sectional area refers to the visible, measurable area of the opening.

FLOW EXPONENT: Flow exponents vary between 0.5 for openings where the flow is dominated by dynamic effects (wide openings, i.e. real orifices) and 1.0 for narrow openings dominated by viscous effects. For the openings associated with infiltration, measurements usually indicate a flow exponent between 0.6 and 0.7

DISCHARGE COEFFICIENT: The discharge coefficient is related to the dynamic effects and is typically close to 0.6 for an orifice and slightly higher for other openings in buildings.

The remaining numbers have little impact on computed flows. Leave the defaults unless you have special requirements.

The three values above describe the flow characteristics of an orifice in typical operation. At extremely low pressure drops the use of the powerlaw equation leads to a division by zero during the network solution process. CONTAMX (& AIRNET) avoids this problem by changing to a linear model in this area. The model is based conceptually on the fact that in nature the flow changes from turbulent to laminar, but the numbers used should not be considered a model of the laminar regime. The hydraulic diameter and Reynolds number are used to determine a point where the model changes from powerlaw to linear.

HYDRAULIC DIAMETER: The hydraulic diameter is equal to $(4 * \text{area} / \text{perimeter})$. For square openings, this equals the square root of the area. For long, thin openings it is twice the width of the opening.

REYNOLDS NUMBER: The transition from turbulent to laminar occurs over a fairly broad range of Reynolds number, being fully laminar below 100.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad. Use this option as you wish, for example, to distinguish between open and closed windows.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Leakage Area → Powerlaw Model

Airflow element name: (required)Description:

Leakage area (choose one)

per item: []
 per unit length: []
 per unit area: []

Reference conditions

Discharge coefficient: []
 Pressure drop: []
 Flow exponent:

use symbol for: small opening
 large opening

quit
 done

[153]

Data describing a "leakage area" is reduced to the powerlaw model:

$$w = C (\Delta P)^n \quad w = \text{mass flow}, \Delta P = \text{pressure drop}$$

Leakage area refers to a description of airflow features given in Chapter 23 of the 1993 ASHRAE Handbook of Fundamentals. This reference includes a large table of typical leakage areas.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

LEAKAGE AREA: Leakage areas in table 3 of Ch 23 are given per item, per unit length, or per unit area, depending on the feature described. Select one of those three descriptions. Number of items, total length, or total area are entered in the path data form.

REFERENCE CONDITIONS: Two sets of reference conditions are common:

discharge coefficient = 1.0 at pressure difference = 4.0 Pa

discharge coefficient = .61 at pressure difference = 10. Pa

Be sure to check the reference conditions for the reported leakage areas.

The flow exponent is not reported and must therefore be estimated. For the openings associated with infiltration, measurements usually indicate a flow exponent between 0.6 and 0.7

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad. Use this option as you wish, for example, to distinguish between open and closed windows.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Connection (ASCOS) → Powerlaw Model

Airflow element name: (required)Description: Flow coefficient: Flow area: []use symbol for: small opening
 large opening quit
 done

Data describing a "connection" is reduced to the powerlaw model:

$$w = C (\Delta P)^n \quad w = \text{mass flow rate, } \Delta P = \text{pressure drop}$$

An exponent of 0.5 is assumed.

The term "connection" refers to the airflow element used in the ASCOS program as described in ASHRAE's "Design of Smoke Management Systems", by Klote & Milke, 1992.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

FLOW COEFFICIENT: The flow coefficient is related to the dynamic effects and is typically close to 0.6 for an orifice and slightly higher for other openings in buildings.

FLOW AREA: The flow area refers to the observable area of the opening.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad. Use this option as you wish, for example, to distinguish between open and closed windows.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Stairwell → Powerlaw Model

Airflow element name: (required)

Description:

Distance between levels: []Cross-sectional area: []Density of people: (pers/m²)Stair treads: open
 closeduse symbol for: ◊ small opening
 ◊ large opening quit
 done

Data describing a stairwell is reduced to the powerlaw model:

$$w = C (\Delta P)^n \quad w = \text{mass flow rate, } \Delta P = \text{pressure drop}$$

An exponent of 0.5 is assumed.

A stairwell will normally be modeled as a vertical series of zones connected by low resistance openings through the floors. The resistance is based on a fit to experimental data presented by Achakji & Tamura.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

DISTANCE: This refers to the distance between openings, normally equal to the distance between levels.

AREA: This refers to the cross sectional (horizontal) area of the stairwell shaft.

PEOPLE: A large number of people on the stairs, as in an evacuation scenario, influences the flow resistance. The experiment used densities of 0, 1, and 2 persons/m². These units are not changeable. (1 m² = 10.76 ft²)

TREADS: Whether the fronts of the stair treads are open or closed also influences the flow resistance.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Ref: Achakji & Tamura, 1988. "Pressure Drop Characteristics of Typical Stairshafts in High-rise Buildings", ASHRAE Transactions.

Shaft → Quadratic Model

Airflow element name: (required)Description: Distance between levels: []Cross-sectional area: []Perimeter: []Roughness: []use symbol for: ◊ small opening
 ◇ large opening quit
 done

Data describing a shaft is reduced to the quadratic model:

$$\Delta P = a w + b w^2 \quad w = \text{mass flow rate, } \Delta P = \text{pressure drop}$$

A shaft will normally be modeled as a vertical series of zones connected by low resistance openings through the floors. The resistance is based on a conduit friction model using the Darcy-Weisbach relation and Colebrook's equation for the friction factor. Ref: ASHRAE, 1993. Handbook of Fundamentals, p 2.8.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

DISTANCE: This refers to the distance between openings, normally equal to the distance between levels. It should also be possible to use this model to represent a horizontal shaft by appropriately redefining distance.

AREA: This refers to the cross sectional area of the shaft.

PERIMETER: The length of the perimeter is used with the area to compute a hydraulic diameter.

ROUGHNESS: This refers to the average size of protrusions from the shaft walls into the airstream.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

[160]

Test Data (1 Point) → Powerlaw Model

Airflow element name: [] (required)

Description: []

Pressure Drop: []

Flow rate: []

Flow exponent: []

use symbol for: [] small opening
[] large opening

[] quit
[] done

[161]

Test data (1 point) is reduced to the powerlaw model:

$w = C (\Delta P)^n$ $w = \text{mass flow rate, } \Delta P = \text{pressure drop}$

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

DATA: The data consists of a single pressure drop and corresponding flow rate. Since this is not enough data to fit both terms in the powerlaw model, you must enter an estimate for the pressure exponent. For the openings associated with infiltration, measurements usually indicate an exponent between 0.6 and 0.7. For large openings an exponent of 0.5 is appropriate.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

[162]

Test Data (2 Points) → Powerlaw Model

Airflow element name: []

Description: []

ΔP : [] Flow: []

ΔP : [] Flow: []

use symbol for: [] small opening
[] large opening

[] quit
[] done

[163]

Test data (2 points) is reduced to the powerlaw model:

$w = C (\Delta P)^n$ $w =$ mass flow rate, $\Delta P =$ pressure drop

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

DATA: The data consists of two different pressure drops and corresponding flow rates.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Test Data (2 Points) → Quadratic Model

[164]

Airflow element name: [] (required)

Description: []

ΔP : [] Flow: []

ΔP : [] Flow: []

use symbol for: [] \diamond small opening
[] \diamond large opening

[] quit
[] done

[165]

Measured data (2 points) is reduced to the quadratic model:

$\Delta P = a w + b w^2$ $w =$ mass flow rate, $\Delta P =$ pressure drop

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

DATA: The data consists of two different pressure drops and corresponding flow rates.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Crack Description → Quadratic Model

Airflow element name: (required)
 Description:

Crack length: []

Crack width: []

Crack depth: []

Number of bends:

quit
 done

Baker, Sharples, and Ward (1987) indicate that infiltration openings can be modeled by a quadratic relationship of the form $\Delta P = a Q + b Q^2$ where Q is the volumetric flow. CONTAM uses the mass flow version of that formula:

$$\Delta P = a w + b w^2 \quad \Delta P = \text{pressure drop, } w = \text{mass flow rate.}$$

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

PARAMETERS: The crack is described in terms of the following four physical characteristics: length, width, distance along the direction of flow (depth), and the number of bends in the flow path.

Note: cracks automatically use the symbol for a small flow path.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Ref: Baker, P.H., Sharples, S., & Ward, I.C. (1987) "Air Flow Through Cracks", Building and Environment, Vol 22, No 4, Pergamon, pp 293-304.

Single Opening with 2-way Flow

Airflow element name: (required)Description: Height: []Width: []Flow coefficient: Minimum ΔT for two-way flow: []use symbol for: \diamond small opening
 \diamond large opening quit
 done

A tall opening may have 2-way flow due to the difference in stack effect at different heights in the opening. This airflow element uses a simple theory to estimate those airflows.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

HEIGHT: This is the distance from the bottom to the top of the opening. Back in the path description, the relative elevation should refer to a point in the center of the opening.

WIDTH: This is the visible width of the opening.

FLOW COEFFICIENT: The default value of 0.78 correlates the experimental data of Weber & Kearney.

MINIMUM ΔT : A two-way flow is driven by the temperature (& air density) difference between the two zones. As ΔT approaches zero, the algorithm approaches a division by zero. To prevent this computational disaster, the two-way model reverts to a one-way powerlaw model using the opening size to define an orifice.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad. For example, use the large opening symbol for an open doorway and the small opening symbol for a closed or nearly closed doorway.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

references:

Brown, W.G. & K.R. Solvason, 1962. "Natural convection through rectangular openings -- 1: vertical partitions", Int. Journal of Heat and Mass Transfer, 5, pp 859-867.

Weber, D.D. & R.J. Kearney, 1980. "Natural convective heat transfer through an aperture in passive solar heated buildings", 5th National Passive Solar Conference, pp 1037-1041.

Two-Opening / 2-way Flow Model

Airflow element name: (required)
 Description:

Height: []

Width: []

Flow coefficient:

use symbol for: ◊ small opening
 ◊ large opening

quit
 done

[171]

A tall opening may have 2-way flow due to the difference in stack effect at different heights in the opening. This airflow element uses two powerlaw model openings at different heights to approximate a single tall opening.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

HEIGHT: This is the distance from the bottom to the top of the opening. Back in the path description, the relative elevation should refer to a point in the center of the opening.

WIDTH: This is the visible width of the opening.

FLOW COEFFICIENT: The default value of 0.78 correlates the experimental data of Weber & Kearney.

SYMBOL: You may select one of two symbols to represent this element and the airflow path of the SketchPad. For example, use the large opening symbol for an open doorway and the small opening symbol for a closed or nearly closed doorway.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Constant Mass Flow Fan Model

[172]

Airflow element name: [] (required)

Description: []

Design (max) flow rate: []

[] quit
[] done

[173]

Data describing a simple constant mass flow rate -- used to move a prescribed amount of air from one zone to another.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

FLOW RATE: Enter the maximum mass flow rate. This value can be modified by the path control profile.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Constant Volume Flow Fan Model

[174]

Airflow element name: [] (required)

Description: []

Design (max) flow rate: []

[] quit
[] done

[175]

Data describing a simple constant volume flow rate fan -- used to move a prescribed volume of air from one zone to another.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

FLOW RATE: Enter the maximum volume flow rate. This value can be modified by the path control profile.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

[179]

The fan performance curve is represented by a cubic polynomial fit to the fan performance data entered here. The performance consists of 4 to 10 pressure rise versus flow rate data points. When both the flow rate and pressure rise are zero, the point is ignored.

The fan performance curve may not include a point of contraflexure (a dip in the curve). If this is indicated by the graph error message, revise some of the pressure rise values in the third column and fit again.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

GRAPH: Graph the data before entering "done".

[180]

Crack Description → Powerlaw Model

Airflow element name: (required)

Description:

Crack length: []

Crack width: []

quit
 done

[181]

Clarke (1985) gives a relationship for flow through cracks that can be converted directly into the powerlaw model.

NAME: A unique name is required for every airflow element for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this airflow element later.

Note: cracks automatically use the symbol for a small flow path.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- an element has not been defined.

Ref: Clarke, J.A., 1985. Energy Simulation in Building Design, Adam Hilger Ltd., Bristol and Boston, p 204.

Simple Source/Sink Model

Source/sink name: (required)
 Description:

Contaminant name:
 Generation rate: []
 Removal rate: []

quit
 done

The general source/sink model uses the following equation:

$$S = G - D * C$$

where S is the contaminant source strength, G is the generation rate, D is the deposition velocity, and C is the current concentration.

NAME: A unique name is required for every source/sink model for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this element later.

CONTAMINANT: The contaminant name must exactly match a name initially defined in the project data. Pressing ← with a blank or the default <...> field will display the contaminant names allowing you to choose one.

GENERATION: A large number of generation units are available; internal units are kg/s.

REMOVAL: The removal rate units are determined from: [D] = [S] / [C] or [kg cont/s] / ([kg cont] / [kg air]) ----> [kg air/s].

For a room filtering device, $G = 0.0$ and $D = f * e$ where f is the flow rate of the room air filter and e is the single pass removal efficiency of the device.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a source/sink model has not been defined.

Pressure Source/Sink Model

Source/sink name: (required)Description: Contaminant name: <reg'd>Generation rate: []Pressure exponent:
 quit
 done

The pressure source/sink model is intended to model contaminant sources which are governed by the inside-outside pressure difference, such as radon or soil gas entry into a basement. In this case the source equation is:

$$S = G \cdot (\Delta P)^n$$

NAME: A unique name is required for every source/sink model for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this element later.

CONTAMINANT: The contaminant name must exactly match a name initially defined in the project data. Pressing ← with a blank or the default <...> field will display the contaminant names allowing you to choose one.

GENERATION: A large number of generation units are available; internal units are kg/s.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a source/sink model has not been defined.

Cutoff Source/Sink Model

Source/sink name: (required)Description: Contaminant name: <req'd>Generation rate: []Cutoff concentration: []
 quit
 done

For volatile organic compounds the source equation is sometimes expressed in the form

$$S = G (1 - C / C_{\text{cutoff}})$$

where S is the contaminant source strength, G is the generation rate, C is the current concentration, and C_{cutoff} is the cutoff concentration at which point emission ceases.

NAME: A unique name is required for every source/sink model for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this element later.

CONTAMINANT: The contaminant name must exactly match a name initially defined in the project data. Pressing ← with a blank or the default <...> field will display the contaminant names allowing you to choose one.

GENERATION: A large number of generation units are available; internal units are kg/s.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a source/sink model has not been defined.

Exponentially Decaying Source Model

[196]

Source model name: (required)

Description:

Contaminant name: <req'd>

Initial emission rate: []

Rate decay constant: []

quit
 done

[197]

For volatile organic compounds the source equation can sometimes be expressed in the form

$$S = G \exp(-t / t_c)$$

where S is the contaminant source strength, G is the initial emission rate, t is the time since the start of emission, and t_c is a decay constant.

NAME: A unique name is required for every source/sink model for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this element later.

CONTAMINANT: The contaminant name must exactly match a name initially defined in the project data. Pressing ← with a blank or the default <...> field will display the contaminant names allowing you to choose one.

EMISSION RATE: The start of emission is controlled by a schedule: When the schedule changes from a zero to a nonzero value, emission begins. The initial emission rate is equal to this value times the schedule value. A single schedule can initiate several emissions at different times.

DECAY CONSTANT: This is the time required for the emission rate to reach 0.37 of the initial rate.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a source/sink model has not been defined.

Boundary Layer Reversible Sink Model

Model name: (required)

Description:

Contaminant name: <req'd>

Film mass transfer coefficient: []

Film density of air: []

Surface mass: []

Partition coefficient:

quit
 done

The boundary layer diffusion controlled reversible source/sink model follows the descriptions presented by Axley.

The rate at which a contaminant is transferred onto the adsorbent is $h d A (C_i - C_s/k)$

where: h = average film mass transfer coefficient over the sink,
 d = film density of air: average of bulk & surface densities,
 A = surface area of the adsorbent,
 C_i = concentration in the air,
 C_s = concentration in the adsorbent,
 k = Henry adsorption constant or partition coefficient.

NAME: A unique name is required for every source/sink model for the search function to work. The name should be descriptive, but since it is short you may also enter a longer description to help you search for this element later.

CONTAMINANT: The contaminant name must exactly match a name initially defined in the project data. Pressing \leftarrow with a blank or the default <...> field will display the contaminant names allowing you to choose one.

COEFFICIENTS: See the references for the estimation of these values. Enter the mass for a unit surface area. Use the source/sink multiplier (screen [100]) as the total area of the adsorbent.

DONE/QUIT: "done" indicates the entries you have made will be saved; "quit" means they will not -- a source/sink model has not been defined.

References:

Axley, J. 1990. "Adsorption Modeling for Macroscopic Contaminant Dispersal Analysis", NIST-GCR-90-573.

Axley, J. 1991. "Reversible Sorption Modeling for Multi-Zone Contaminant Dispersal Analysis", Building Simulation - 91, International Building Performance Simulation Association.

APPENDIX D: CONTAM File Formats

D.1 The Weather File

The weather file, necessary for transient simulations, must be created with an editor. The file name is determined by the user; the file name extension, .WTH, is required.

The first line of the weather file is a user specified description to help identify the use of the file. The second line gives the number of contaminants. It is followed by the contaminant names, one per line. This contaminant data is used by CONTAM to verify that the file is appropriate to the building description. The remaining data lines consist of the time in hour:minute:second format, the ambient temperature [K], the absolute barometric pressure [Pa], the wind speed [m/s], the wind direction [degrees clockwise from north], and the mass ratio for each contaminant [$\text{kg}_{\text{cont}} / \text{kg}_{\text{air}}$].

Consider the following example with data for two contaminants:

```
weather for testccl; T=20, const; cont = C1, C2.  
2  
C1  
C2  
00:00:00 293.15 101325. 0.0 0.0 0.0 0.0  
24:00:00 293.15 101325. 0.0 0.0 0.0 0.0
```

D.2 Spreadsheet Files

Spreadsheet files record the transient data for all contaminants in a single zone in a for suitable for input into a spreadsheet program. See data screens [143] and [146] for the creation of spreadsheet files.

The first two lines of data are a series of column labels, enclosed in quote marks and separated by commas. The first line consists of two "time" labels followed by the names of all contaminants. The second line gives the units of time and contaminant concentration. On the remaining lines, one for each listing time step (screen [122]), is the time in hour:minute:second format, the time in seconds, and the concentration of each contaminant.

Consider the following example with data for two contaminants:

```
"time", "time", "C1", "C2"  
"of day", "[s]", "[ppm]", "[ppm]"  
"00:00:00", 0, 105.821, 105.821  
"00:15:00", 900, 96.5156, 96.5156  
"00:30:00", 1800, 87.9715, 87.9715  
.  
.  
.  
"23:45:00", 85500, 115.945, 115.945  
"24:00:00", 86400, 105.821, 105.821
```

Note that the file does not identify the zone, level, or project file. You must keep track of these separately in conjunction with the user defined file name.

