NIST Technical Note 1887

CONTAM User Guide and Program Documentation Version 3.2

W. Stuart Dols

Brian J. Polidoro

This publication is available free of charge from:
http://dx.doi.org/10.6028/NIST.TN.1887

NIST National Institute of Standards and Technology • U.S. Department of Commerce

NIST Technical Note 1887

CONTAM User Guide and Program Documentation Version 3.2

W. Stuart Dols

Brian J. Polidoro

Engineering Laboratory

This publication is available free of charge from:
http://dx.doi.org/10.6028/NIST.TN.1887

September 2015

U.S. Department of Commerce
Penny Pritzker, Secretary

National Institute of Standards and Technology
Willie May, Under Secretary of Commerce for Standards and Technology and Director

NIST National Institute of Standards and Technology • U.S. Department of Commerce

ABSTRACT

This manual describes the computer program CONTAM version 3.2, developed by NIST. CONTAM is a multizone indoor air quality and ventilation analysis program designed to help determine airflows, contaminant concentrations, and personal exposure in buildings. Airflows include infiltration, exfiltration, and room-to-room airflow rates and pressure differences in building systems, and can be driven by mechanical means, wind pressures acting on the exterior of the building, and buoyancy effects induced by temperature differences between zones, including the outdoors. Contaminant concentrations include the transport and fate of airborne contaminants, due to airflow, chemical and radio-chemical transformation, adsorption and desorption to building materials, filtration, and deposition to and resuspension from building surfaces. Personal exposure includes the exposure of building occupants to airborne contaminants, for eventual risk assessment.

CONTAM can be useful in a variety of applications. Its ability to calculate building airflow rates and relative pressures between zones of the building is useful for assessing the adequacy of ventilation rates in a building, for determining the variation in ventilation rates over time, for determining the distribution of ventilation air within a building, for estimating the impact of envelope air-tightening efforts on infiltration rates, and for evaluating the energy impacts of building airflows. The program has also been used extensively for the design and analysis of smoke management systems. The prediction of contaminant concentrations can be used to determine the indoor air quality performance of buildings before they are constructed and occupied, to investigate the impacts of various design decisions related to ventilation system design and building material selection, to evaluate indoor air quality control technologies, and to assess the indoor air quality performance of existing buildings. Predicted contaminant concentrations can also be used to estimate personal exposure based on occupancy patterns.

Version 2.0 contained several new features including: non-trace contaminants, practically unlimited number of contaminants, contaminant-related libraries, separate weather and ambient contaminant files, building controls, scheduled zone temperatures, improved solver to reduce simulation times and several user interface related features to improve usability. Version 2.1 introduced more new features including the ability to account for spatially varying external contaminants and wind pressures at the building envelope, more new control elements, particle-specific contaminant properties, total mass released calculations and detailed program documentation. Version 2.4 introduced two new deposition sink models, a one-dimensional convection/diffusion contaminant model for ducts and user-selectable zones, new contaminant filter models, control super nodes, super filters, a duct balancing tool, building pressurization and model validity tests and several other usability enhancements.

Version 3.0 added a deposition with resuspension source/sink model, a self-regulating vent airflow model, and integrated coupling between Computational Fluid Dynamics (CFD) and multizone modeling. Version 3.1 introduced a variable time step ordinary differential equation solver (VODE) for solving the contaminant equations and coupling between the TRNSYS energy simulation program and CONTAM to enable combined energy, airflow and contaminant transport analysis. Version 3.2 added an ultra-violet germicidal irradiation (UVGI) filter model, contaminant exfiltration result files, a SQLite database result file, and co-simulation between the EnergyPlus whole building energy analysis program and CONTAM.

Key Words: airflow analysis; building controls; building technology; co-simulation; computer program; contaminant dispersal; controls; energy use; indoor air quality; multizone analysis; smoke control; smoke management; ventilation

DISCLAIMER

CONTAM

This software was developed at the National Institute of Standards and Technology by employees of the Federal Government in the course of their official duties. Pursuant to title 17 Section 105 of the United States Code this software is not subject to copyright protection and is in the public domain. CONTAM is an experimental system. NIST assumes no responsibility whatsoever for its use by other parties, and makes no guarantees, expressed or implied, about its quality, reliability, or any other characteristic. We would appreciate acknowledgement if the software is used. This software can be redistributed and/or modified freely provided that any derivative works bear some notice that they are derived from it, and any modified versions bear some notice that they have been modified.

Users are warned that CONTAM is intended for use only by persons competent in the field of airflow and contaminant dispersal in buildings and is intended only to supplement the judgement of the qualified user. The computer program described in this report is a prototype methodology for computing the airflows and contaminant migration in a building. The calculations are based upon a simplified model of the complexity of real buildings. These simplifications must be understood and considered by the user.

Certain trade names and company products are mentioned in the text or identified in an illustration in order to adequately specify the equipment used. In no case does such an identification imply recommendation or endorsement by the National Institute of Standards and Technology, nor does it imply that the products are necessarily the best available for the purpose.

SUNDIALS (CVODE Solver)

Copyright © 2002-2014, Lawrence Livermore National Security.

Produced at the Lawrence Livermore National Laboratory.

All rights reserved.

This file is part of SUNDIALS.

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice, this list of conditions, and the disclaimer below.
2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions, and the disclaimer (as noted below) in the documentation and/or other materials provided with the distribution.
3. Neither the name of the LLNS/LLNL nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission.

DISCLAIMER: THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL LAWRENCE LIVERMORE NATIONAL SECURITY, LLC, THE U.S. DEPARTMENT OF ENERGY, OR ITS CONTRIBUTORS BE

LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Additional BSD Notice

1. This notice is required to be provided under our contract with the U.S. Department of Energy (DOE). This work was produced at Lawrence Livermore National Laboratory under Contract No. DE-AC52-07NA27344 with the DOE.
2. Neither the United States Government nor Lawrence Livermore National Security, LLC, nor any of their employees, makes any warranty, express or implied, or assumes any liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately-owned rights.
3. Also, reference herein to any specific commercial products, process, or services by trade name, trademark, manufacturer or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or Lawrence Livermore National Security, LLC. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or Lawrence Livermore National Security, LLC, and shall not be used for advertising or product endorsement purposes.

National Institute of Standards and Technology Technical Note 1887

Natl. Inst. Stand. Technol. Tech. Note 1887 (August 2015)

CODEN: NSPUE2

ACKNOWLEDGEMENTS

The authors would like to acknowledge the development of the original CONTAM program by George N. Walton (retired) and James W. Axley during their time at NIST. The Computational Fluid Dynamics capabilities were developed by Liangzhu (Leon) Wang of Concordia University while he was a graduate student at Purdue University and a guest researcher at NIST. The CVODE solver was integrated into CONTAM by David Lorenzetti of the Lawrence Berkeley National Laboratory, who also developed the simple trust region method of the Newton-Raphson non-linear airflow solver during his time as a postdoctoral research associate at NIST.

Some of the features of the program were sponsored by other agencies including the following:

- Version 2.4. Development of many features were sponsored by the Naval Surface Warfare Center (NSWC) Dahlgren Division under Military Interdepartmental Purchase Request (MIPR) N00178-05-MP-00139. The authors appreciate the input of the Kathrina Urann and Matthew Wolski of NSWC. John Goforth of the Lawrence Livermore National Laboratory provided input on developing the socket communications capability of ContamX to enable co-simulation with the Analytical Conflict and Tactical Simulation tool.
- Version 3.1. The Defense Threat Reduction Agency (DTRA) sponsored LBNL's work on the VODE solver that was performed under U.S. Department of Energy contract number DE-AC02-05CH11231. NIST was supported by the NSWC Dahlgren Division under MIPRs N0017810MP00069 and N0017810MP00160. We would like to acknowledge Matthew Wolski of NSWC for his continued interest and direction in furthering the capabilities of the CONTAM program.
- Version 3.2. UVGI filters, SQLite database output and direct scheduling via continuous value and discrete value files was sponsored by the Pennsylvania State University under agreement number E12-072071. NSWC, under the direction of Matthew Wolski, also supported the development of the contaminant exfiltration tracking capabilities and finalization of the release of this version under MIPR N0017814MP00108.

DOCUMENT REVISION HISTORY

The table below provides a summary of revisions made to the documentation since the initial release on October 7, 2005. You can use the links in the Section column of the table to "jump" to the revised section of the document.

Date	Version	Description	Section
2015-06-08	3.2	EnergyPlus-CONTAM coupling	Working with EnergyPlus
	3.2	Added Penn State UVGI filter model	Penn State UVGI Filter
	3.2	Added contaminant exfiltration and sqlite result files	Results Files
	3.2	Added direct scheduling via files	CVF and DVF Controls
2012-12-04	3.1	TRNSYS-CONTAM coupling with Type98	Working with TRNSYS
	3.1	Program/project configuration	Configuring ContamW
	3.1	Simulation time step evaluation and CVODE solver	Simulation Run Control
2011-08-09	3.0a	Added self-regulating vent airflow element	Self-regulating Vent Model
2010-12-14	3.0	Added Deposition with Resuspension source/sink element	Source/sink Elements
	3.0	Added CFD capabilities	Zone Properties, Airflow Path Properties, Simulation Properties
	3.0	Changed name of ContamX2.log file to <prj name>.xlog and relocated to same directory as the prj file.	Zone Properties, Airflow Path Properties, CSM File
2008-10-06	2.4c	Modified CSM file format	XLOG File
	2.4c	Open Input/Output files	Support Files
2008-04-25	2.4c	New mathematical control elements	Control Element Types
2008-04-03	2.4b	Display/hide airflow results lines	Results Display
2008-02-28	2.4b	Create WPC File Dialog Box	Create WPC File Dialog
	2.4b	Recent file list	Working with Project Files
	2.4b	Open Project Input/Output Files from ContamW	Working with Project Files
	2.4b	Changed location of program log file storage	Getting Help, Components of a CONTAM Project
	2.4b	Socket message details	Appendix E
2006-09-08	2.4b	Zone occupancy sensor	Sensors
	2.4b	Simulation parameters – density control	Airflow Numerics Properties
	2.4b	Review transient wind pressure	Checking Wind
	2.4b	Network deletion (ducts and controls)	Deleting Building Component Icons
2006-07-17	2.4b	Sensor time constant	Control Element Types
2006-06-06	2.4b	Equations clarified for control elements that implement Gain and Scale.	Control Element Types

Date	Version	Description	Section
2006-05-10	2.4b	New source sinks based on NRCC emission data base	Source/Sink Elements
	2.4b	Super source/sink element	Super Source/Sink Element
	2.4b	Gage pressure sensors	Control Element Types
	2.4b	Air mixture data corrected	Non-trace Contaminants and Water Vapor
	2.4b	One-dimensional zone axis orientation	1D Zone Data
	2.4b	Enhanced SketchPad deletion capabilities	Deleting Building Component Icons
	2.4b	Results display time step navigation	Viewing Results
	2.4b	Results navigation jump setting	Results Display
	2.4b	Maximize ContamW window on startup	SketchPad Size
	2.4a	Short time step simulation results files	Results Files and Output Properties
	2.4a	Use of WPC and DVF files to control airflow calculation	Weather Properties
2005-11-14	2.4	Duct terminal design flow rates	Terminal Properties
	2.4	Transient simulations using steady state weather	Run Control Properties
	2.4	Units of display in the status bar	Status Bar
2005-10-20	2.4	WPC File format version update 2.4	WPC File Format

TABLE OF CONTENTS

Abstract ..... **iii**
Disclaimer ..... **iv**
Acknowledgements ..... **vi**
Document Revision History ..... **vii**
1 Introduction **1**
1.1 Program Modifications 2
1.2 System Requirements 9
2 Getting Started **10**
2.1 Installing CONTAM 10
2.2 Running CONTAM 10
2.3 User Tasks 11
2.4 The CONTAM Graphical User Interface 13
2.5 Components of a CONTAM Project 15
3 Using CONTAM **19**
3.1 Working with the SketchPad 19
3.2 Configuring ContamW 33
3.3 Working with Levels 38
3.4 Working with Zones 41
3.5 Working with Airflow Paths 46
3.6 Working with Simple Air-handling Systems 66
3.7 Working with Ducts 73
3.8 Working with Controls 91
3.9 Working with Species and Contaminants 111
3.10 Working with Sources and Sinks 117
3.11 Working with Filters 129
3.12 Working with Kinetic Reactions 137
3.13 Working with Occupant Exposure 141
3.14 Working with Data and Libraries 144
3.15 Working with Weather and Wind 149
3.16 Working with WPC Files 157
3.17 Working with Schedules 165
3.18 Working with Simulations 169
3.19 Working with Results 185
3.20 Working with Project Annotations 218
4 Special Applications of CONTAM **219**
4.1 Building Pressurization Test 219
4.2 Smoke Control Systems 220
4.3 Shafts 220
5 Working with TRNSYS **222**
5.1 TRNSYS-CONTAM Utilities and Setup 222
5.2 Software Components and Files 222
5.3 TRNSYS-CONTAM Coupling Process 223
6 Working with EnergyPlus **233**
6.1 EnergyPlus-CONTAM Coupling Process 235
6.2 EnergyPlus Input Files 238
6.3 Getting Help 240
7 Theoretical Background **241**
7.1 Model Assumptions 241
7.2 Contaminant Analysis 243
7.3 Airflow Analysis 253
8 References **270**
Appendix A - PRJ File Format **275**
Appendix B - ContamX TCP/IP Socket Communication **309**

1 INTRODUCTION

CONTAM is a multizone indoor air quality and ventilation analysis computer program designed to help you determine:

- *Airflow Rates*: infiltration, exfiltration, and room-to-room airflow rates in building systems driven by mechanical means, wind pressures acting on the exterior of the building, and buoyancy effects induced by the indoor and outdoor air temperature difference.
- *Contaminant Concentrations:* the dispersal of airborne contaminants transported by these airflows; transformed by a variety of processes including chemical and radio-chemical transformation, adsorption and desorption to building materials, filtration, and deposition and resuspension to and from building surfaces, etc.; and generated by a variety of source mechanisms.
- *Personal Exposure*: the prediction of exposure of occupants to airborne contaminants for eventual risk assessment.

CONTAM can be useful in a variety of applications. Its ability to calculate building airflow rates and relative pressures between zones of the building is useful for assessing the adequacy of ventilation rates in a building, for determining the variation in ventilation rates over time, for determining the distribution of ventilation air within a building, and for estimating the impact of envelope air-tightening efforts on infiltration rates, and for evaluating the energy impacts of building airflows. The program has also been used extensively for the design and analysis of smoke management systems [Klote, et al. 2012]. The prediction of contaminant concentrations can be used to determine the indoor air quality performance of buildings before they are constructed and occupied, to investigate the impacts of various design decisions related to ventilation systems and building material selection, to evaluate indoor air quality control technologies, and to assess the indoor air quality performance of existing buildings. Predicted contaminant concentrations can also be used to estimate personal exposure based on occupancy patterns.

This document addresses both the graphical user interface (referred to herein as **ContamW**) and the numerical solver (referred to herein as **ContamX**) of the program, collectively referred to as **CONTAM**.

1.1 PROGRAM MODIFICATIONS

Program Enhancements

CONTAM 3.2 is the latest version in the line of CONTAM programs. The history of major enhancements is provided below. The latest version of CONTAM is backwards compatible with previous versions, meaning you can open existing project files created with previous versions. Once updated to the latest version of the program, project files can not be saved as or opened with previous versions. Therefore, you should maintain older copies of your existing project files.

Throughout this manual you will find new features (latest revision) of the program highlighted as illustrated by this paragraph.

Version 3.2

EnergyPlus-CONTAM Multizone Airflow, Heat Transfer and Contaminant Analysis - This capability includes modifications to ContamX and the creation of a new dynamic link library (DLL) that implements co-simulation between EnergyPlus and ContamX via the Functional Mockup Interface (FMI). This includes the development of ContamFMU.dll and enhancing CONTAM3DExport.exe to enable the creation of EnergyPlus input files for the purposes of performing co-simulation.

Contaminant Exfiltration Result Files - Added the ability to create result files that provide the mass of contaminants that are transported to ambient.

SQLite Database Result Files - Added the ability to create a SQLite database file (.sqlite3) that contains the same results as the simulation results file (.sim).

UVGI Filter - Added an ultra-violet germicidal irradiation filter element.

Direct Scheduling via CVF and DVF Controls - Added the ability to directly schedule items, e.g., source/sinks and airflow paths, via external files without an associated controls networks drawn on the SketchPad.

Version 3.1

TRNSYS-CONTAM Multizone Airflow, Heat Transfer and Contaminant Analysis - This capability includes modifications to ContamW and ContamX and the creation of a new TRNSYS Type and the software utilities to provide a new coupling process that enables ContamX to be more directly coupled with TRNSYS Type56.

ContamW Pseudo-geometry - Provide ability to define scaling factor for drawing on the ContamW SketchPad and view wall dimensions on the SketchPad. This feature is an aid to utilities provided for the TRNSYS-CONTAM coupling that create three-dimensional representations for TRNSYS Type56 multizone heat transfer calculation.

ContamW Temperature Plots - Provide ability to plot temperatures in ContamW.

ContamX - Added socket communication messages that provide for the transfer of temperatures and airflow to/from ContamX.

Type98 - Acts as a bridge between ContamX and Type56 for TRNSYS simulations.

Utilities - Created software utilities *CONTAM3DExport.exe* to extrude scaled CONTAM prj representations to form *Trnsys3d idf* files and *Type56-98Coupler.exe* to form the linkage between Type56 and Type98.

CVODE - Implemented a variable time step solver for transient contaminant transport calculations. CVODE, is a general-purpose code for solving ordinary differential equations, that provides for variable time steps, high-order integration methods, and automatic error control.

Usability Enhancements

Particle Plots - Provide ability to plot particle distributions when simulating multiple size ranges with ContamW.

Result Mode Toolbar - Provide toolbar buttons to manipulate result time step display.

Simulation Time Step Analysis - Provide ability for user to check schedules and input files (WTH, CTM, DVF, CVF) for maximum time step required to capture all scheduled events.

Program/Project Configuration - Modified ContamW configuration to separate out Program-related options from Project-related options.

Result File Output - Provide user with control over output of simulation result files CSM, LOG, and SRF.

Resolved Program Issues

Restart File - Restart (RST) file did not properly account for Deposition/Resuspension source/sinks. Also, modified restart file behavior to only write restart data at the end of each day (24:00:00) and/or at the end of the last day if shorter than 24 hours. User is no longer require to select the time to use when utilizing the restart file, only the date.

PLD File and Duct Terminals - Duct terminals were not being output to the Path Location Data (PLD) file.

Results Display Window and Duct Leakage - The results display window was not providing duct leakage airflow results for zones having junctions in them with associated duct leakage.

Version 3.0a

Self-Regulating Vent - A new airflow element has been added to CONTAM. The self-regulating vent limits the airflow rate in both directions through a flow path with user-defined limiting air pressure differences. CONTAM implements the self-regulating vent as described by Axley 2001 Residential Passive Ventilation Systems: Evaluation and Design.

Surface Node Results File - Added new results file (SRF) for Boundary Layer and Deposition/Resuspension source/sink elements.

New Zone Color Option - Now when revealing the level below there is an option to see the zone colors of the zones on the level below.

Resolved Program Issues

ContamW

Fix - crash could occur when deleting a level.

Fix - enable transient simulation times that span the Dec/Jan transition, i.e., allow simulations to start before December and end after January.

Fix - when replacing a wind pressure profile with the Find & Replace dialog the wall azimuth and wind speed modifier should be set to the defaults.

Fix - allow plotting of results for a simulation that spans the Dec/Jan transition.

Fix - crash occurred when deleting zones having occupancy schedules that referenced them.

Fix - incorrect painting of the toolbar for some Windows themes.

Fix - cubic spline airflow element curves were not plotted correctly if their curves were invalid.

Modification - consolidated variable density airflow calculation parameters.

ContamX

Fix - Matrix reordering issue when very large project files encountered fatal errors when matrix reordering was attempted due to size of index parameter.

Fix - Prevent intermittent program crash when running under Windows 7 64 bit OS.

Fix - Contaminant summary file (CSM) issues: values when working with non-trace boundary layer diffusion and deposition/resuspension source sinks were likely too high due to summation of the source contributions over multiple iterations of the variable density iterations within each time step; combinations of non-trace and trace contaminants could result in trace contaminant-related summations not being accounted for. However, these issue did not affect sim file results.

Fix - Issue related to non-trace contaminant calculations involving Boundary Layer Diffusion and Deposition/Resuspension sources whereby the contribution of mass added to the zone in which these source/sinks were located was incorrectly calculated.

Fix - Issues related to Deposition with Resuspension source/sink element when performing variable density calculations.

Fix - Power Law source/sink model was not utilizing the correct source/sink element function.

Fix - Enable DVF files to handle Dec/Jan transition.

Fix - Issues related to Decaying (exponential decay) source elements: generation rate was not reset when scheduling the source on/off, and the generation rate was being compounded when multiple Decaying sources were being used.

Modification - Improved variable density calculations.

Version 3.0

Coupled Multizone/CFD - Implemented the ability to simulate an internal zone using computational fluid dynamics (CFD). Details of this capability are provided via the NIST Multizone Modeling website.

Deposition/Resuspension Source/Sink Element - Added a new source/sink element that accounts for both deposition and resuspension of particle contaminants to/from a surface.

Windows 7 64-bit - ContamX crashed intermittently due to data copy error.

Usability Enhancements

Editing Context Menu - Right mouse button to provide cut, copy and paste functions.

Version 2.4c

New control types - Added new mathematical functions: exp, ln, log10, etc.

TCP/IP socket simulation control - Enabled multi-day simulation control.

Operating System Compatibility

CONTAM Fonts - ContamW SketchPad now utilizes internal bitmap patterns instead of font files.

HTML Help - The ContamW help system has been converted to HTML help for compatibility with the Windows Vista operating system.

CONTAM Log Files - The contamx2.log and contamw2.log files are now stored in the users application directory (home directory for Linux version of ContamX).

Usability Enhancements

ContamX Non-Convergence Information - If the ContamX airflow calculation fails to converge, a list of airflow nodes that failed to converge will be provided in the contamx2.log file.

Error List - ContamW generates a list of problem icons that is displayed in the newly implemented *Building Check Problems* dialog box when a PRJ file does not pass the building check performed prior to a simulation. This includes airflow paths that do not have valid location data when working with WPC files.

Create WPC File Dialog - The WPC dialog box has been modified to enable "manual" generation of PLD files to aid in the creation of WPC files, and the Weather page of the Simulation Parameters property sheet has been modified accordingly.

Recent file list - Added list of recently opened project files to the File menu.

Open Project Input/Output Files - Provided the ability to open CONTAM-related input and output files from the File menu. Files are opened based on file extension associations of Windows.

Result Plotting - Right click on SketchPad icons when viewing results to activate result plotting.

SketchPad Navigator - Display a small icon SketchPad window to simplify navigation of the main SketchPad with larger icons: from main menu select View - SketchPad Navigator.

Duct Results Display - duct velocity calculated based on upstream air density.

CSM File Format - modified CSM file format to provide more detail related to source/sinks.

Resolved Program Issues

Cubic Spline Elements - Enabled the use of cubic spline elements in duct segments.

CSM File - WPC file contaminants filtered by envelope flow paths were not being accounted for in the Contaminant Summary File.

DVF and WTH File - WTH file records were being skipped when using both a DVF and a WTH file.

Duct Balancing - After duct balance is performed the flow rates of "mass flow type" fans could be incorrectly set as though they were volume flow types.

Flow Limits - Flow limits of airflow paths were set back to none upon file read.

CVF/DVF Node Selection - Node drop down list was improperly sorted causing potential for setting of incorrect nodes.

Version 2.4b

Building Controls

Occupancy Sensor - Added zone occupancy sensor type.

Gage Pressure Sensor - Added gage pressure sensors for zones, duct junctions and terminals.

Source/Sink Elements

NRCC Sources - Added powerlaw and peak contaminant source types based on NRCC material emission database.

Super Source/Sinks - Implemented ability to combine multiple source/sink elements into a single super source/sink element.

Usability Enhancements

Enhanced deletion capabilities - Provided ability to delete multiple items from the SketchPad with a single deletion operation.

Results display - Improved control over selection of simulation time for results display.

Wind pressure display - Provided ability to review transient wind pressures resulting from wind data provided by WPC files.

SketchPad dragging - Hold down the mouse wheel to drag the SketchPad when scroll bars are visible.

Library Manager - Use the Ctrl and Shift keys to select multiple items from the element lists and perform copy and delete operations on them.

Version 2.4a

Number of building components - The limit on the total number of each type of building component (i.e., zones, paths, ducts, sources/sinks, etc.) that can be created within a single project has been increased from 32 767 to 2 147 483 647.

Short time-step method - A new simulation mode that solves contaminant dispersal equations using an explicit solution method.

1D Zones - The ability to simulate selected zones as one-dimensional convection/diffusion zones has been implemented. This provides the ability to more realistically simulate the transport of contaminants through "long" or "tall" zones, e.g., hallways and shafts, better accounting for transport times through these 1D zones.

1D duct model - Implementation of one-dimensional convection/diffusion elements in ductwork, thus providing more realistic contaminant transport results through the duct system. This requires the use of the short time-step method.

Duct temperature calculations - When simulating 1D ducts in the short time-step mode, CONTAM can now calculate duct temperatures based on the "mixing" of air streams at the duct junctions and the source (zone) air temperatures from which air is introduced into the duct system.

Simulation results data - Modified the *zone* and *junction pressures* as reported to the simulation results file to more closely resemble gage pressures. Zone pressures are now referenced to the ambient pressure at the level on which the zone is located. Junction static pressure is now referenced to the zone in which the junction is located at the height of the junction within the zone.

Automated duct balancing - Added simulation option to automatically balance duct systems. This will greatly simplify the task of defining detailed duct systems in CONTAM.

Building model verification tests

Building pressurization test - Simulation option to automatically determine building envelope airtightness based on a simulated fan pressurization test.

Building airflow test - Simulation option that generates a set of data, mostly related to building ventilation, that can be used to gauge the reasonableness of model inputs before beginning analysis of a building.

Cubic spline airflow elements

Contaminant Filters

New filter models - CONTAM has added *particle* and *gaseous* filter models that greatly increases the user-s flexibility to create models based on measured filter performance data, e.g., MERV and breakthrough curves.

Filter super elements - enables multiple filter models to be combined into a single filter element, e.g., a particle pre-filter combined with a gaseous filter.

Contaminant summary file - This file is generated when simulating contaminants to provide information related to source/sink contaminant generation and removal, filter loading and breakthrough, and contaminant transport between building zones and ambient.

Deposition sink models - Added Deposition Velocity and Deposition Rate sink models to simplify the definition of sinks based on more familiar deposition terminology.

Control super elements - The task of creating building control systems has been improved by reducing the amount of user input required, increasing the amount of flexibility in defining control systems and enabling the sharing of control elements, e.g., sensors, within and between projects.

TCP/IP socket simulation control - ContamX can be controlled via the TCP/IP Socket communication protocol based on a pre-defined set of messages that include control commands and data exchange. ContamX can also be compiled and run under Linux based operating systems, and a Linux-compatible executable is now available on the NIST web site.

Usability enhancements

Find - search for items on SketchPad by Name and Number

Find-and-replace - search for and replace properties of building components within a project

SketchPad movement
Ctrl + arrow key => skip 10 cells in direction of arrow
Shift + arrow key => move to next icon in the direction of arrow
Mouse wheel for scrolling:

Wheel => up/down Shift+Wheel => page up/down

Alt+Wheel => left/right Shift+Alt+Wheel => page left/right

Keyboard icon placement - use keyboard to place icons on the SketchPad

Floating status bar

User-defined *zone colors*

User-defined *duct colors* and *automated duct tracing/coloring tool*

Generate *bitmap file* (.bmp) of SketchPad

Enhancements from version 2.0 to 2.1

Wind pressure and contaminant fields - The ability to incorporate data from exterior airflow and pollutant transport models, e.g., plume and puff dispersion models, to utilize detailed ambient wind pressure and contaminant data fields to provide boundary conditions on the airflow paths of the envelope of built structures

Control elements - New control elements to simulate time delays associated with spin-up/down of fans and the opening/closing of dampers and to perform integration, peak determination of sensor output over time, maximum, minimum and exponential operations

Particle analysis - Modified contaminant properties to simplify the analysis of airborne particles

Mass release calculation - The calculation of total mass released by contaminant sources during a simulation

Program Documentation - Programming documentation of the software was produced which can be obtained from the NIST website. This documentation includes details of CONTAM input and output file formats [Walton and Dols 2003].

Enhancements from version 1.0 to 2.0

Building controls - Controls include sensors, actuators, modifiers and links. Control actuators can be used to modify various characteristics of building components based on control signals obtained from sensors and even modified by signal modifiers. For example, a sensor can be used to obtain a contaminant concentration within a zone, and a proportional control actuator can be used to adjust supply airflow into the zone based on the sensed concentration.

Scheduled zone temperatures - Zone temperatures can now be varied through the use of user-defined schedules. This allows for the change in zone pressures due simply to the change in temperature within the zone according to the ideal gas relationship.

Contaminants

Non-trace contaminants - You can now account for the impact of contaminant concentrations on the density of the air, e.g., water vapor.

Unlimited number of contaminants - CONTAM no longer restricts the number of contaminants you can simulate. The previous limitation was 10.

Contaminant-related libraries - Contaminant related elements can now be shared through CONTAM library files. These elements include contaminant species, filters, source/sinks and kinetic reactions.

Numerics

Variable air density - CONTAM now provides the ability to simulate non-flow related processes that can lead to the accumulation/reduction of mass within building zones, e.g., due to non-trace contaminant sources and to variations in the zone pressure due to the change in zone temperature.

Sparse matrix techniques implemented to greatly reduce transient simulation times for large problems.

Separated solver from graphical user interface to provide for batch execution of simulations and directly utilize .PRJ files.

Transient weather

Separate transient weather and contaminant files - Weather files (WTH) no longer contain contaminant concentrations (except for outdoor humidity ratio). This means you don-t have to create different weather files depending on the types of contaminants you are simulating. CONTAM now provides you with the option of simulating transient ambient contaminant concentrations using a contaminant file (CTM).

Weather file creation/conversion software - NIST has developed a software tool that allows you to convert existing weather files to CONTAM 2.0 compatible weather files. You can convert your existing 1.0-compatible files, TMY2 and EnergyPlus weather files.

User interface

Longer zone names - Zone names can now be up to 15 characters long.

SketchPad zooming feature - You can now reduce the icon size of the SketchPad to allow the display of larger projects on the screen.

Results Display

Net inter-zone airflows displayed for highlighted zones in Results Display Window

Airflow direction indicators are now displayed in the Status Bar when viewing airflow path results

Distinct simple air-handling system zones - The implicit zones of multiple simple air-handling system are now distinguished from each other to allow for the plotting of individual system zones

1.2 System Requirements

The CONTAM interface (ContamW) runs under Windows XP, Vista, 7 and 8. However, the CONTAM simulation engine (ContamX) can run under both Windows and Linux environments, and both versions are provided on the CONTAM download website (See Installing CONTAM).

2 GETTING STARTED

2.1 INSTALLING CONTAM

Obtaining CONTAM

CONTAM installs from a set of installation files that you can obtain from NIST. These files can be downloaded from the NIST website (www.bfrl.nist.gov/IAQanalysis).

Installing CONTAM

After downloading CONTAM from the NIST website, double-click the Microsoft installer file (.MSI) to begin the installation process. Choose a folder into which you want to install the program, or simply select the default location which is "C:\Program Files\NIST\CONTAM XX" (where XX refers to the version number).Follow the directions to complete the installation. This installation will not remove or overwrite previous versions of CONTAM.

Uninstalling CONTAM

The CONTAM setup program also provides you with an uninstall feature. You uninstall CONTAM much as you would a typical Windows program. Access the Control Panel from the Settings selection of the Start menu. Select Add/Remove Programs from the Control Panel. Select CONTAM XX from the list of installed programs and click the "Add/Remove" button to uninstall CONTAM.

2.2 RUNNING CONTAM

As indicated in the Installing CONTAM section, CONTAM actually consists of two executable programs: ContamW and ContamX. ContamW is the graphical user interface and ContamX is the simulation engine. Typically you will activate ContamX via a menu command from within ContamW. However, ContamX can be utilized as a command-line tool and even utilized within script or batch files.

Running ContamW

Run ContamW by selecting CONTAM ##(where ## is the version number, e.g., 3.2) from the NIST program group of the Start menu.

Running ContamX as a Command-line Program

The easiest way to run ContamX as a command-line program is by locating the contamx3.exe file and related projects and supporting files within a common directory. The command-line format for running ContamX is as follows:

CONTAMX3 <project filename>

If you leave off the project file name, you will be prompted to enter one, therefore when running in batch mode be sure to provide a filename.

Running ContamX with TCP/IP Socket Communication

ContamX can be run in a TCP/IP socket communication mode that enables run-time control over the simulation including execution control and data transfer via TCP/IP sockets. See Appendix B for details and a complete description of the TCP/IP messaging structure.

2.3 USER TASKS

The use of CONTAM to analyze airflow or contaminant migration in a building involves five distinct tasks:

1. *Building Idealization*: Form an *idealization* or specific *model* of the building being considered,
2. *Schematic Representation*: Develop a schematic representation of the idealized building using the ContamW *SketchPad* to draw the building components,
3. *Define Building Components*: Collect and input data associated with each of the building components represented on the *SketchPad*,
4. *Simulation*: Select the type of analysis you wish to conduct, set simulation parameters, and execute the simulation,
5. *Review & Record Results*: Review the results of your simulation and record selected portions of the results.

Task 1 - Building Idealization

Building idealization refers to the simplification of a building into a set of zones that are relevant to the user's goal in performing an analysis. A building can be idealized in a number of ways depending on the building layout, the ventilation system configuration and the problem of interest. This idealization phase of analysis requires some engineering knowledge on the part of the user and is an acquired skill that you can develop through experience in airflow and indoor air quality analysis and by becoming familiar with the theoretical principles and details upon which indoor air quality analysis is based.

It is important to note that CONTAM provides a macroscopic model of a building. In this macroscopic view, each zone is considered to be well-mixed. Well-mixed means that a zone is characterized by a discrete set of state variables, i.e., temperature, pressure and contaminant concentrations. Temperature and contaminant concentration do not vary spatially within a zone, and contaminants mix instantly throughout well-mixed zones. However, pressure does vary hydrostatically within all zones.

Beginning with CONTAM version 2.4, one-dimensional convection/diffusion zones can be implemented within CONTAM. This feature can be useful in simulating contaminant transport through long or tall zones that are characterized by a single, dominant flow direction. One dimensional convection/diffusion duct models can also be implemented to more realistically capture contaminant transport within entire duct systems.

CONTAM is well suited for analyzing the interaction between the zones of a building on a *macroscopic* or whole-building level. Computational Fluid Dynamics (CFD) analysis is well-suited for analyzing the airflow and contaminant transport characteristics on a *microscopic* level, i.e., within a single zone of a building. However, the computational resources required to perform a CFD analysis for an entire building is currently prohibitive. Beginning with CONTAM version 3.0, a zero-order turbulence CFD model has been directly integrated with CONTAM to enable consideration of a single, CFD zone in conjunction with the rest of a building that is being analyzed on a macroscopic level, i.e., coupled multizone/CFD analysis.

Task 2 - SketchPad Representation

Developing the SketchPad representation will be the focus of your interaction with ContamW. With ContamW's SketchPad you will be able to draw a diagram – a *SketchPad diagram* – of your building idealization using drawing tools and sets of icons to represent components of the building system. CONTAM translates your diagram into a system of equations that will than be used to model the behavior of the building when you perform a simulation.

See Working with the SketchPad in the Using CONTAMW section.

Task 3 Data Entry

Data entry can be one of the more time-consuming parts of the process of using ContamW. It involves the determination and input of the numerical values of the parameters associated with each of the SketchPad icons. These icons represent the elements of the building model and include air leakage paths (windows, doors, cracks), ventilation system elements (fans, ducts, vents), contaminant sources, filters, and sinks and control network components. Each of these elements is associated with a number of parameters, and you must obtain the values of these parameters for entry into the model. Depending on the element and the application, these values can be obtained from building-specific data, engineering handbooks, and product literature. In many cases, a degree of engineering judgement will be involved. ContamW allows you to create libraries of these elements that you can use in current and future modeling efforts.

Detailed information is provided for the various components throughout the Using CONTAMW section.

Task 4 Simulation

Simulation is the use of CONTAM to solve the system of equations assembled from your SketchPad representation of a building to predict the airflow and contaminant concentrations of interest. This step involves determining the type of analysis that is needed; steady state, transient or cyclical, and a number of other simulation parameters. These parameters depend on the type of analysis you wish to perform (steady state or transient), and include convergence criteria and in the case of a transient analysis, time steps and the duration of the analysis.

See Working with Simulations in the Using CONTAMW section.

Task 5 Review & Record Results

ContamW allows you to view the simulation results on the screen and to output them to a file for input to a spreadsheet program or other data analysis programs including those developed by NIST and available on the CONTAM website (e.g., SimRead2 and ContamRV). Airflows and pressure differences at each flow element can be viewed directly on the SketchPad. Contaminant concentrations for a zone can also be plotted as a function of time directly from the SketchPad. You can then decide which data you wish to examine more closely and export these to a tab-delimited text file that can then be imported into a spreadsheet for further analysis. There is also a controls-related feature that provide the ability to report the values of user-selected control nodes to a control "log" file for each time step of a transient simulation.

See Working with Results in the Using CONTAMW section.

2.4 THE CONTAM GRAPHICAL USER INTERFACE

ContamW is the graphical user interface (GUI) of CONTAM. Use ContamW to create and view your airflow and contaminant dispersal analysis projects. It consists mainly of a drawing region referred to as the SketchPad, a set of drawing tools, a title bar, a set of menus, and a status bar. The following sections provide brief explanations of each of the features of the CONTAM GUI. See the Using ContamW section for details on how to use these features.

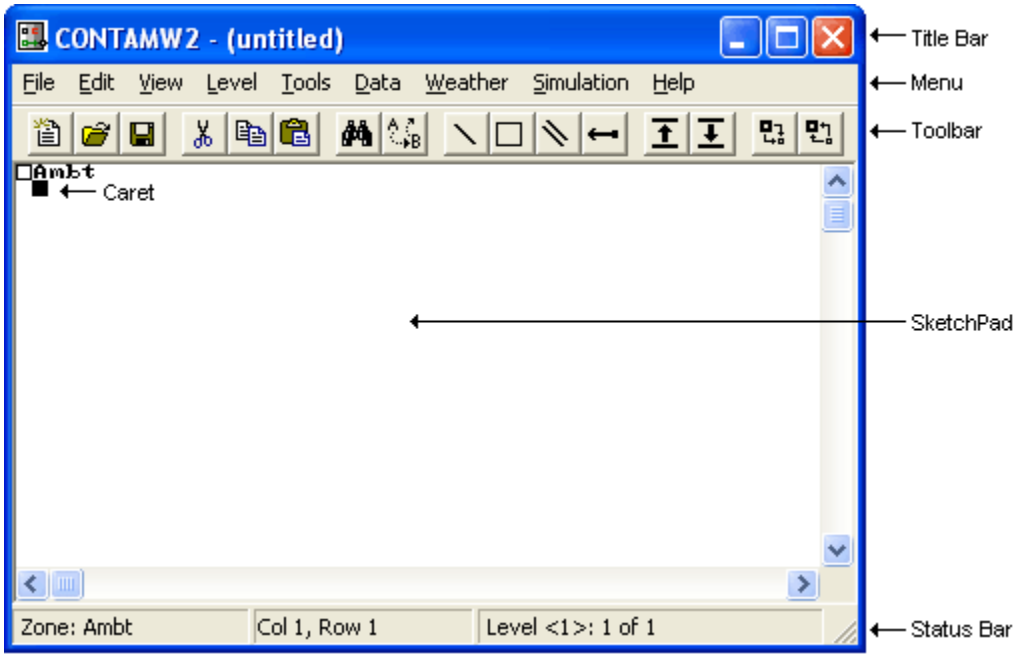


Figure - The CONTAM Graphical User Interface

2.4.1 Title Bar

The title bar is the typical rectangular region at the top of the main ContamW window. The CONTAM project filename will be displayed within this region.

2.4.2 Menu

The menu is typical of a Windows program with differences that provide functionality specific to the CONTAM application. It is through this menu that most ContamW operations can be performed including: saving and retrieving project files, selecting various modes of display, setting up and performing simulations, as well as accessing the on-line help system. Note that some of the menu items have shortcuts or hot-keys that enable quick access; for example, to save the current project file use the Ctrl+S key combination.

2.4.3 Toolbar

The toolbar, shown in the following figure, appears below the menu and provides convenient shortcuts to some of the menu items. Several of the toolbar buttons are similar to those found in other Windows applications. Other buttons provide a shortcut to functionality specific to ContamW.

2.4.4 Status Bar

The status bar, shown in the following figures, is the region displayed below the SketchPad at the bottom of the main window. This region is broken up into three separate panes that display various information depending on the current mode of the SketchPad.

ContamW 2.4 now provides you with a floating status bar which will display in the region of the caret whenever you select certain icons on the screen providing a more convenient means to review icon information. This is similar to Tool Tips that appear when you hover over a toolbar button with the Windows cursor. You can turn this feature on and off via the **View → Floating Status Bar** or **Ctrl+T** for Tool tips.

Left Pane

This pane always displays the type and number of building component icon, e.g., zone, path, air handling system, etc.

In the *normal mode* of operation, the leftmost pane displays summary information of the currently highlighted cell or icon.

In the simulation *results mode*, the leftmost pane displays the results for the currently highlighted icon. For a zone this includes the temperature and pressure relative to the ambient pressure at the elevation of the level on which the zone is located. For duct junctions and terminals it displays the temperature and static pressure relative to the zone in which the junction or terminal is located. For paths and duct segments it will display the airflow and pressure drop across the path or segment along with symbols to indicate the direction of flow (>, <, ^, v). For a simple air handling system icon the outdoor airflow, recirculation airflow and exhaust airflows for the implicit flow paths will be displayed.

Many units of display can be controlled by selecting the SI or IP system of units in the Project Configuration Properties accessible via the **View →Options...** menu command.

Center Pane

In the *normal mode* of operation, the center pane indicates the location of the currently highlighted icon or cell in SketchPad coordinates (numbered from the top-left corner). In the simulation *results mode* the center pane displays the current simulation time step for which the results are being displayed.

Right Pane

The rightmost pane displays the name and number of the current level and the total number of levels in the project.

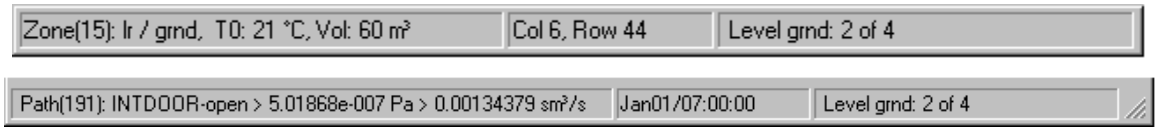


Figure - Status Bar during Normal Mode and Results Viewing Mode respectively

2.5 Components of a CONTAM Project

2.5.1 Project Files

All data related to the characteristics of the projects you work with are stored in a "project" file having a "PRJ" extension (See Working with Project Files). This is an ASCII file, which is intended to be "viewed" only by the ContamW program. You should keep careful records of your project files and establish a naming convention that is meaningful to you for the various versions of a project that you may wish to save. There are other files utilized by CONTAM including: simulation results files, ambient files, library files, log files and other support files.

Results Files

Simulation results are stored in files created automatically by ContamX with the same name as the PRJ file except that the "PRJ" extension is replaced by extensions that depend on the type of results generated by the simulation (see Working with Simulation Results).

Ambient Files

Ambient files include weather (WTH extension), contaminant (CTM extension) and wind pressure and contaminant files (WPC extension). Ambient files may contain up to a years worth of data and are used when performing transient simulations (see Working with Weather). These are ASCII files, but the formats are unique to CONTAM.

Library Files

Library files are the means by which you can share various types of data between CONTAM projects (see Working with Data and Libraries).

Support Files

Support files include log files, backup files and a configuration file. ContamW support files are created in the user's application directory - c:\Documents and Settings\<user>\Application Data\NIST\ContamW\. The ContamX log file is created in c:\Documents and Settings\<user>\Application Data\NIST\ContamX\. The Linux version of ContamX stores the ContamX log file in the user's home directory. Log files contain information about the execution of the respectful programs and are useful in the event that you require technical support from the program developers (see Getting Help). ContamW creates a backup (BKP extension) file every time a PRJ file is successfully opened. The configuration file (CFG extension) and the BKP file are also stored in the ContamW application directory.

2.5.2 Building Components

Building components are the items that characterize the physical makeup of a building that you define using ContamW. This section briefly describes these components.

Levels

CONTAM represents buildings in terms of multiple levels, accounting for the communication of air and contaminants between these levels. Levels typically correspond to floors of a building, 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.

Walls

Walls are used to designate zones which are regions surrounded by walls, floor and ceiling. These walls include the building envelope and internal partitions with a significant resistance to airflow.

Floors and Ceilings

Floors and ceilings are included implicitly by ContamW for building zones. When you draw a zone on the SketchPad, ContamW automatically includes the floor of the zone. To create a roof with penetrations into the floor below requires a blank level above the top floor. It is also possible to create a phantom zone with no floor or ceiling as might be required to create an atrium that spans multiple levels (see Working with Zones).

Zones

In CONTAM, a zone typically refers to a volume of air having a uniform temperature and contaminant concentration. However, beginning with version 2.4, zones can now be configured as one-dimensional *convection/diffusion zones* in which contaminant concentrations are allowed to vary along a user-specified axis. Beginning with version 3.0 a single zone or region of a building can be defined as a CFD zone.

There are three types of zones in CONTAM: *normal*, *phantom* and *ambient*. *Normal zones* are separated from the zone below by a floor. The *ambient zone*, which surrounds the building is implicitly defined and is identified by the symbol at the upper-left corner of the SketchPad. You can also use an *ambient zone* icon to define a courtyard. *Phantom zones* indicate that the area on the current level is actually part of the *normal zone* on the level immediately below. There is no floor between a *phantom zone* and the *normal zone* below. You could use *phantom zones* to define building features such as atriums. Only *normal zones* can be configured to be *convection/diffusion* zones, however, these zones can still be referenced by *phantom zones*.

Airflow Paths

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 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. CONTAM can implement several different models or airflow elements to define airflow paths. The basic categories of airflow elements or models are as follows: small and large crack/openings represented by power-law and quadratic pressure relationships, small and large doorways elements, and fan/forced airflow elements. (See Working with Airflow Paths)

Simple Air-handling Systems

The simple air-handling system (AHS) provides a simple means of introducing an air-handling system into a building without having to draw a duct system. It provides a reasonable model of an air-handling system that delivers user-specified flows where the system is properly balanced and the fan is not impacted by any other pressurizing effects in the building. The AHS consists of two implicit airflow nodes (return and supply), three implicit flow paths (recirculation, outdoor, and exhaust air), and multiple supply and return points that you place within the zones of the building. You can set the air flows of the AHS to vary according to a schedule.

Ducts

You can use ducts to implement a more detailed model of an air-handling system that handles a broader range of conditions. For example, when an air handler is off, the ductwork may provide flow paths between zones which are significant in relation to the normal construction cracks or openings. Ductwork consists of duct segments (paths) and junctions or terminal points (nodes). CONTAM can implement several different duct segment models or duct flow elements to define duct segments. The basic categories of duct flow elements are as follows: resistance models, fan performance curves, and back-draft dampers. (See Working with Ducts)

Contaminants, Sources and Sinks

You can define an unlimited number of contaminants within a single project with a practically limitless number of sources associated with the contaminants. CONTAM can simulate contaminant transport via airflow between zones, removal by filtration mechanisms associated with flow paths, and removal and addition by chemical reaction. CONTAM can also implement several source and sink models to generate contaminants within or remove contaminants from a zone. These models include: constant generation, pressure driven, decaying source, cutoff concentration, reversible boundary layer diffusion, and burst models. (See Working with Contaminants and Working with Sources and Sinks)

Schedules

Schedules are used to control or modify various properties of building components as a function of time. You can set schedules for airflow paths, duct flow paths; contaminant sources and sinks; and inlets, outlets and outdoor air delivery of simple air-handling systems. The effect of setting a schedule on a building component varies depending on the properties of the component. For example, you can set a schedule to adjust the airflow delivered to a zone by an inlet of a simple air-handling system. (See Working with Simple Air Handling Systems) CONTAM also provides the ability to schedule zone temperatures.

Controls

Controls include sensors, actuators, modifiers and links. Control actuators can be used to modify various characteristics of building components based on control signals obtained from sensors and even modified by signal modifiers. For example, a sensor can be used to obtain a contaminant concentration within a zone, and a proportional control actuator can be used to adjust supply airflow into the zone based on the sensed concentration.

Occupants

Occupants can be used to determine the amount of contaminant exposure a person would be subjected to within a building. Occupants can also generate contaminants. You can set a schedule to establish each occupant's movement within a building. Occupant schedules can also be used to define periods of times when occupants are not in the building. (See Working with Occupant Exposure)

2.5.3 Weather

CONTAM enables you to account for either steady-state or varying weather conditions. Weather conditions consist of ambient temperature, barometric pressure, humidity ratio, wind speed and direction, as well as ambient contaminant concentrations.

2.5.4 Simulation

In CONTAM, simulation is the process of forming a set of simultaneous equations based upon the information stored in the project file, performing the numerical analysis to solve the set of nodal equations according to user-defined specifications, and creating simulation results files that can be viewed using the ContamW interface. There are three basic types of simulations that you can perform for airflow and contaminant analysis using CONTAM: steady state, transient and cyclical. (see Working with Simulations)

3 USING CONTAM

This section provides detailed information on how to use the features of the CONTAM program as well as a detailed explanations of the terminology of the user interface.

3.1 WORKING WITH THE SKETCHPAD

The SketchPad is the region of the ContamW screen where you draw the schematic representation of a building you wish to analyze. This representation is in the form of a set of simplified floor plans that represent the levels of a building. The SketchPad consists of an invisible array of cells into which you place various icons to form your schematics of a building. The SketchPad is used to establish the geometric relationships of the relevant building features and is not necessarily intended to produce a scaled representation of a building. It should be used to create a simplified model where the walls, zones, and airflow paths are topologically similar to the actual building.

Pseudo-Geometry

While ContamW does not require you to draw building floor plans to scale, the SketchPad does provide the ability to assign a scale value to the SketchPad cells. You can choose to display this so-called *pseudo-geometry* via the Project Options of the ContamW Configuration. This will provide dimensions in the status bar as you draw on the SketchPad. It will also provide you with scaled information for walls, zones and ducts. It will provide wall areas when you highlight wall icons, which can be useful when defining multipliers of airflow path icons associated with the wall. When the pseudo-geometry mode is activated, default values will be provided for zone volume (based on scaled floor area) and duct lengths. In order to distinguish between pseudo-geometry values and actual values as saved in the prj file, scaled values will be enclosed in "{}" when presented in the SketchPad Status Bar. This feature is intended to simplify the coupling with energy analysis software, e.g., TRNSYS and EnergyPlus, which require geometric information in order to perform energy related calculations (See Working with TRNSYS).

The Caret

When working with ContamW you will notice a blinking square on the SketchPad. This is known as the system *caret*, and it is the size of a single SketchPad cell. This *caret* is the same thing as the blinking vertical bar that is common to many word processing applications. The *caret* indicates the currently selected cell of the SketchPad. Any icon-related information that appears in the status bar is associated with the location of the *caret*.

Moving the Caret

To move the *caret* around the SketchPad you can use the keyboard arrow keys or you can move the system cursor with the mouse and click the *left mouse button* to set the caret position. You can also use the arrow keys in conjunction with the **Shift** and **Ctrl** keys to move the *caret* more quickly.

Shift + Arrow Key moves the *caret* 10 cells in the direction of the arrow. The number of cells to move can be modified within the ContamW Configuration (See Cell/Icon Size in the Configuring ContamW section).

Ctrl + Arrow Key moves the *caret* to the next icon in the direction of the arrow

SketchPad Operations

The specific operations that you will perform using the SketchPad are as follows:

1. Drawing Walls, Ducts and Controls
2. Drawing building component icons
3. Defining building component icons
4. Viewing results
5. Viewing envelope pressure differentials due to wind effects

SketchPad Modes

There are three basic modes of the SketchPad: ***normal***, ***results*** and ***wind pressure***. The SketchPad mode basically refers to the type of information that is displayed upon the SketchPad. You can tell what mode the program is in by looking at the items in the **View** menu to see which one has a mark next to it.

- In the *normal* mode ContamW displays only the building component icons. In this mode you can add, delete, copy, and move icons.
- In the *results* mode, ContamW displays simulation results upon the SketchPad. **In this mode you will not be allowed to add, delete, copy and move icons upon the SketchPad** (See Viewing Results).
- The *wind pressure* mode is provided to verify wind speed and direction information visually on the SketchPad (See Checking Wind Pressure Data).

Control Super Element / Super Node SketchPad

With the advent of the new control super element, comes another use of the SketchPad to create and edit super elements and super nodes (See Control Super Elements in the Working with Controls section). When working with super elements only the controls drawing tool is enabled, along with the ability to define the control network icons. This SketchPad is activated via the **Data→Controls→Super Elements…** menu item. When the Super Element SketchPad is active, the upper left corner of the SketchPad will display "Super Element:" followed by the name of the super element currently displayed on the SketchPad. When working with super elements the "Control Super Elements" dialog box will also be displayed. The Super Node SketchPad enables only the modification of existing control sub-node icons, so the control drawing tool will be disabled as will be the ability to delete control network icons. The Super Node SketchPad is activated by instantiating an existing control super element or double-clicking on a Super Node icon. When the Super Node SketchPad is active, the upper left corner of the SketchPad will display "Super Node:" followed by the name of the super node currently displayed on the SketchPad. When working with super nodes the "Super Node Data" dialog box will also be displayed (See Control Super Nodes).

Zooming the SketchPad

You can change the size of the cells in which icons appear on the SketchPad in order to zoom the floor plan in and out on the SketchPad. The amount of information that will appear on your screen is dependent on the resolution and the cell size. Zooming can be accomplished using the two zoom buttons on the toolbar and their associated keyboard shortcuts shown below or changing the Current Cell/Icon Size on the Cell/Icon Size page of the Project Configuration Properties dialog box accessible via the **View→Options...** menu selection (see Cell/Icon Size).

Keyboard Shortcuts:

Ctrl + PageUp to increase cell size

Ctrl + PageDown to decrease cell size

Printing SketchPad Images

You can obtain images of your SketchPad drawings to print or edit using the Windows print screen feature. To do this, size the ContamW window and press **Alt+PrintScrn** on the keyboard to copy the current window to the Windows clipboard. Then you can immediately paste the image into the desired program. For example, you can paste the image into the Windows Paint program for editing or directly into a word processing program. You can then print the image from either of these programs.

Exporting .BMP SketchPad Files

You can save a SketchPad image of the currently displayed level to a Windows bitmap (.BMP) file using the **File→Save SketchPad to .BMP File...** menu item. The Save As dialog box will appear allowing you to name the file. The file will contain the entire SketchPad based on the current cell/icon size and color settings.

3.1.1 Drawing Walls, Ducts and Controls

You use various tools to draw the physical features of the building such as walls and ducts. All drawing can be accomplished with either the mouse or the keyboard. When drawing, the following mouse and keyboard keys produce common behavior. The left mouse button (**LMB**) corresponds to the ↵, or **Enter** key; the right mouse button (**RMB**) to the **Esc** key. Vertical motion of the cursor corresponds to the ↑ and ↓ cursor keys, and horizontal motion corresponds to the ← and → cursor keys.

The basic steps for drawing walls and ducts are as follows:

1. Activate drawing tool,
2. Set the initial location of the object,
3. Draw the object,
4. Undo current drawing,
5. Finalize drawing the object, and
6. Deactivate drawing tool.

Activating a Drawing Tool

There are four drawing tools available, two for drawing walls, one for drawing ducts and one for drawing control networks. To activate a drawing tool either select it from the toolbar, select it from the "Tools" menu, or use the shortcut associated with the menu item. The toolbar button of a selected drawing tool will remain depressed as long as that tool is active. The toolbar buttons and associated menu shortcuts are shown here.

rectangle (or box) drawing tool (**Ctrl+B**)

free-form wall drawing tool (**Ctrl+W**)

duct drawing tool (**Ctrl+D**)

controls drawing tool (**Ctrl+L**)

Once you have selected the tool, the drawing cursor will be displayed. The drawing cursor is a pink square the size of a single SketchPad cell. Initially, the cursor appears with a transparent center.

Setting the Initial Location of the Object

To begin drawing the object, you set the initial location by first moving the drawing cursor either with the mouse or the keyboard arrow keys and then you either click the **LMB** or press ↵ on the keyboard. This will anchor the beginning of the object at the nearest valid SketchPad cell for the type of object you are drawing. When you select a valid beginning location, the drawing cursor will become solid and you can begin drawing your object.

Drawing the Object

After you anchor the beginning of the object, you simply use the mouse and/or keyboard arrow keys to draw the desired shape. While you are drawing, the cursor will be restricted to the SketchPad region and constrained to specific movements in order to maintain drawing within valid cells of the SketchPad. As you move the cursor, a dark line will appear on the SketchPad representing the shape of the object you are drawing.

You can also move more quickly using the **Shift + Arrow Key** and **Ctrl + Arrow Key** keyboard combinations.

Shift + Arrow Key moves the drawing cursor 10 cells in the direction of the arrow. The number of cells to move can be modified within the ContamW Configuration (See Cell/Icon Size in the Configuring ContamW section).

Ctrl + Arrow Key moves the drawing cursor to the next icon in the direction of the arrow, which may be a wall, duct or control.

Undoing the Current Drawing

Prior to finalizing the drawing of the object, you can undo your drawing. To undo what you have drawn, either single-click the **RMB** or press the **Esc** key. This will erase the thick dark line, but you will still be in the drawing mode. To begin drawing again, set the initial location again and continue drawing another object.

Finalizing the Drawing Object

Once you are satisfied with your drawing, you finalize the object by either single-clicking the **LMB** or pressing the ↵ key. This will replace the cells that had the thick dark line with the icons appropriate for completing the type of object you are drawing. If you have attempted to draw an invalid object, you will be prompted with a dialog box containing a message indicating the type of error, and then you will be allowed to repair the drawn object.

Deactivating Drawing Tools

When you are finished using a drawing tool you click the **RMB** or press **Esc** until your system cursor reappears. Also, selecting a different drawing tool will automatically deactivate the current tool if you do not currently have an object anchored.

3.1.2 Drawing Building Component Icons

A set of icons is used to represent various building components such as airflow paths (representing doors, windows, cracks, etc.), contaminant sources and sinks, occupants and air-

handling systems. You ***place icons on the SketchPad using the Right Mouse Button*** to display a pop-up menu and then selecting the desired building component from the menu. You can also use the **Tools → Draw Icon** menu and associated keyboard shortcuts to place building icons that can be place with the pop-up menu.

The icon placement menus are context-sensitive. The menu selections that you can choose from the icon placement menus will depend on the contents of the cell occupied by the caret and whether the SketchPad is currently displaying simulation results. ***When results are being displayed, you will not be allowed to place additional icons on or delete icons from the SketchPad.*** This is done to prevent the display of misleading results on the SketchPad due to a mismatch in the number of icons on the SketchPad and the number of icons for which results are available from the last simulations. The icon placement menu will also prevent you from placing icons on invalid SketchPad locations. For example, you cannot place a supply or return of a simple air-handling system within the ambient zone.

Icon	Keyboard Shortcut
Zone	Z
Ambient	A
Phantom	P
Flow Path	F
Air Handling System	H
Supply (Inlet)	I
Return (Outlet)	O
Source/Sink	S
Exposure	X
Annotation	N

Table - Keyboard shortcuts for placing icons on the SketchPad

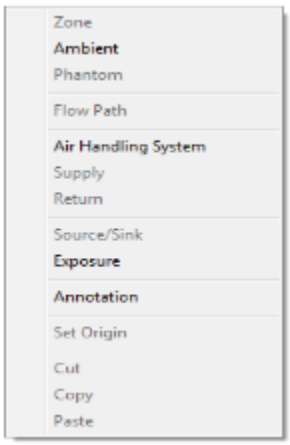


Figure - Icon placement menu (right-click on SketchPad)

The following table is a list of the icons that you will see on the ContamW SketchPad. This list shows the icons by categories of building components. Some of these icons are placed on the SketchPad using the drawing tools, and others are placed using the pop-up icon placement menu.

Icon Category	Component Icons
Walls	
Zones	
Duct Segments	
Duct Junctions	
Duct Terminals	
Simple AHS	
Airflow Paths	
Source/Sinks	
Occupants	
Controls	

Table - CONTAM SketchPad Icons (shown in default colors)

3.1.3 Defining Building Component Icons

When a new building component icon is drawn upon the SketchPad, it appears as a red icon. The color red indicates that the parameters of the component must still be defined. To define a building component, you double-click (**LMB**) on the icon or move the caret to the icon and press the **Enter** key. This will activate the associated dialog box or property sheet for the selected component, into which you enter the parameters that define the component. After you have entered the parameters of the component, click the **OK** button to accept the parameters or the **Cancel** button not to accept the parameters. The detailed properties of each component icon will be presented throughout the "Using ContamW" section of this manual.

NOTE - Press the **F1 Help** key to activate the *context-sensitive help system* that explains the properties of each building component.

3.1.4 Deleting Building Component Icons

There are two general methods of deleting building components from the SketchPad: one-at-a-time (single component) or in groups (multiple components). CONTAM must be in the *normal* SketchPad mode in order to delete icons.

Single Component Deletion

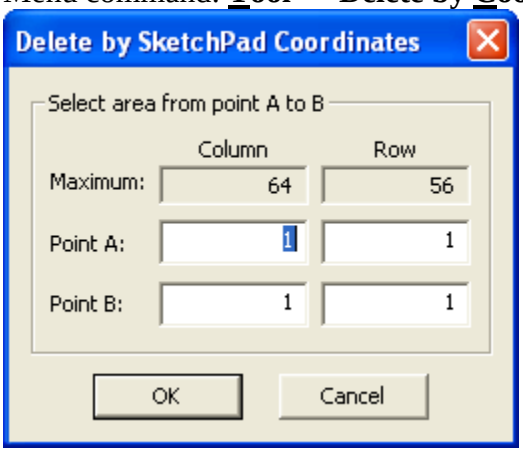
To delete individual building components, you simply highlight the icon by either clicking (**LMB**) on the icon or moving the caret to the icon using the arrow keys then either press the **Delete** key or use the **Edit** → **Delete** menu command.

Multiple Component Deletion

ContamW provides the ability to delete multiple building components in a single deletion operation. This is done by first using one of three methods to select a region on the SketchPad then activating the **Edit** → **Delete** menu command (or pressing the **Delete** key).

Use one of the following three deletion tools to select a region of the SketchPad to work with:

- Delete by region
  Menu command: **Tool** → **Delete by Region**
  Tool bar button:
- Delete by box:
  Menu command: **Tool** → **Delete by Box**
  Tool bar button:
- Delete by coordinates
  Menu command: **Tool** → **Delete by Coordinates**



Then press **Delete** to display the Multiple Icon Deletion Options dialog box shown below. Use the check boxes to select the types of icons you want to delete, select the level(s) from which you want to delete the icons using the Ctrl and Shift keys to select multiple levels from the list, then click the OK button to perform the deletion.

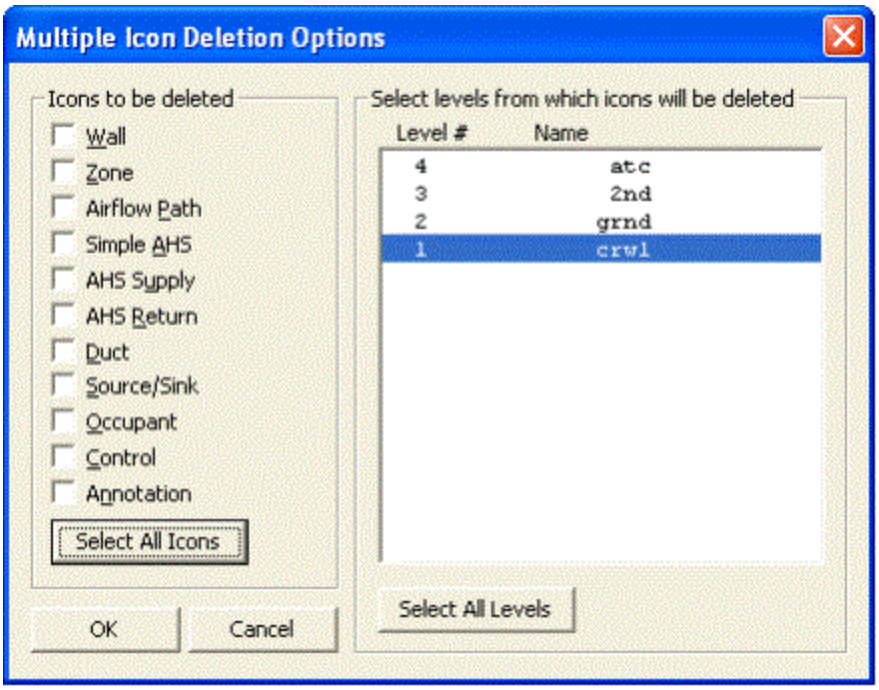


Figure - Multiple Icon Deletion Options dialog box

NOTE: Some deletion operations will invalidate existing relationships between items in the project file. This can lead to the inability of ContamW to save the project file and/or ContamX to perform a simulation on the project file. For example, deleting a zone that is referenced by an occupant schedule will prevent ContamW from saving the project file. However, an error message will be provided to notify you when you attempt to save. For this reason, ***you should always create a backup copy prior to performing major deletion operations.***

Network Deletion (Ducts and Controls)

You can delete a set of connected ducts or controls using a single command. Highlight any icon within the network you wish to delete and select **Edit → Delete Network** from the menu or use the **Ctrl+Delete** keyboard shortcut. If deleting a control network, you may be prompted whether to delete or undefine phantom nodes that reference the network being deleted.

3.1.5 Find and Replace Building Component Data

ContamW provides the ability to perform both simple searches for building components and to perform find-and-replace operations. Both features involve the use of a dialog box that can be displayed simultaneously with the SketchPad to allow interaction between both the SketchPad and the dialog boxes.

Find

Use the Find feature to locate various building component icons on the SketchPad by either their *Number* or *Name* properties. Be aware that numbers may change whenever the total number of each type of component is modified by either adding or deleting components. Whenever the project file is saved, the components will be renumbered based on their SketchPad position starting from the top level and working from left to right then top to bottom on the SketchPad down through the building levels.

Menu Command: **Edit → Find…**
Keyboard Shortcut: **Ctrl+F**

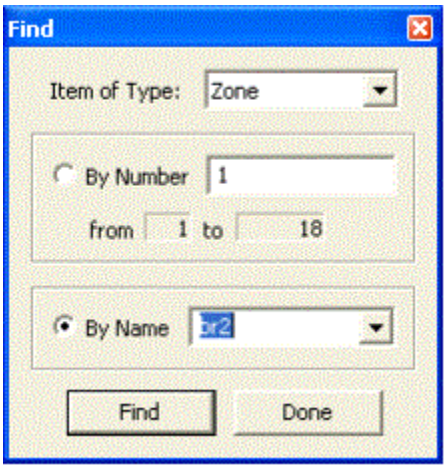


Figure – Find dialog box

Simply select the type of component icon you wish to locate, select either a number or name from the lists provided and click the *Find* button. The icon will be located and highlighted on the SketchPad. If there are no components of a given type, the search controls will be disabled.

Searchable items:

- Air Handling Systems
- Airflow Paths - including Supplies and Returns of Simple Air Handling Systems
- Control Nodes - sub-nodes of super nodes will defer to the super node
- Duct Junctions
- Duct Segments
- Occupants
- Source/Sinks
- Zones - including implicit supply and return zones of simple air handling systems

Find and Replace

The *Find and Replace* feature consists of a more advanced *find* feature that enables you to generate lists of items based on user-selected ranges of property values. You can then replace parameters of the items in the list of items found.

Menu Command: **Edit → Replace…**
Keyboard Shortcut: **Ctrl+R**

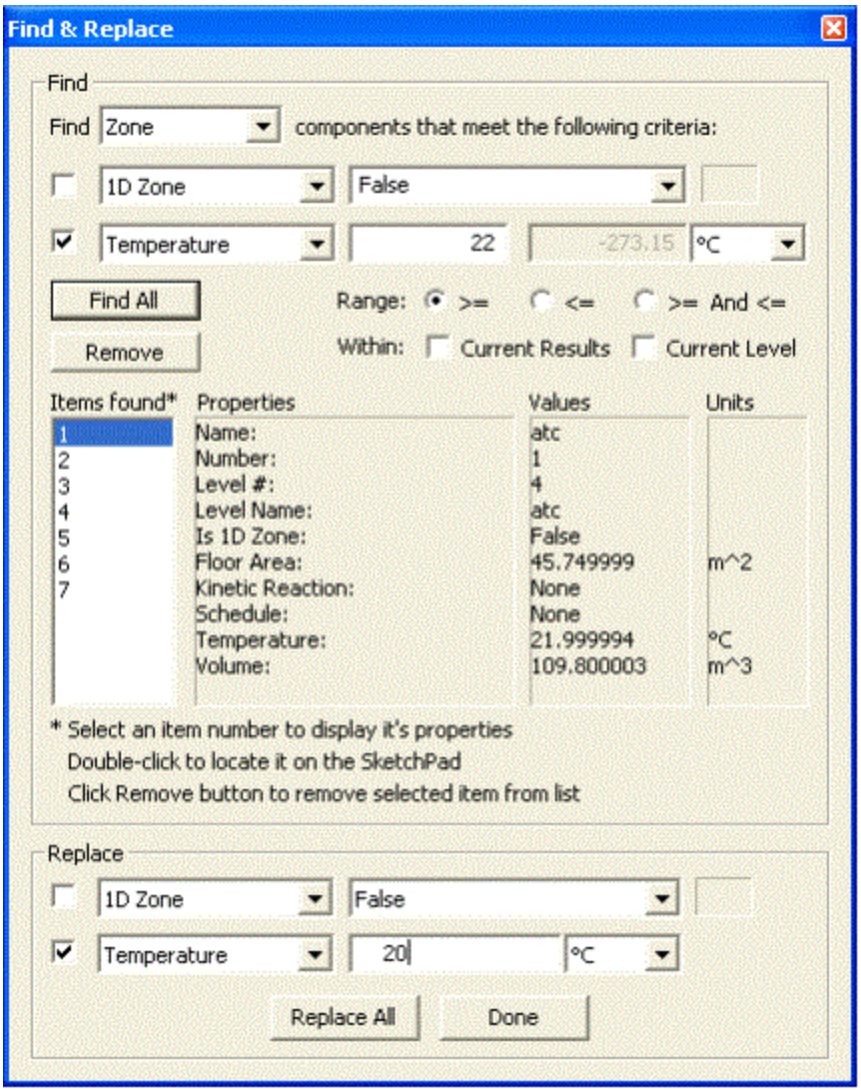


Figure – Find and Replace dialog box

There are two sections of the Find and Replace dialog box: Find and Replace. Use the dialog box controls within the Find section to generate a list of items that match the desired search criteria. Once you have generated a list of items found, then select the items within the Replace section to replace the desired parameters of the items within the list. You are not required to use the replace feature; you can simply use this tool to locate specific items within a project file.

Each type of building component has two predefined sets of parameters upon which searches can be performed: list-based and value-based. The list-based search parameters are accompanied by a list of each parameter type contained in the project, e.g., elements, schedules, filters, etc. You can either select from the list to search for an exact match or type in a case-sensitive search string of text to match all items that contain the search string. Value-based parameters allow you to search for items based on a range of values of the selected parameter. It is possible to search using either of these sets of parameters or both. Use the check boxes that are located adjacent to the search parameters to select which ones to use.

Once you have set your search criteria, click the *Find All* button to initiate the search. This will generate a list of *Items found*. You can then select items in the list to display a summary of their properties, or double-click to locate the item on the SketchPad. You can also remove individual items from the list by selecting the item and clicking the *Remove* button. This allows you to refine the set of items for replacement.

Building Component	List Parameters	Value Parameters
Air Handling System	Kinetic reaction in return Kinetic reaction in supply Outdoor air filter Return air filter Outdoor air schedule	Minimum outdoor airflow Return system volume Supply system volume
Air Handling System Supply and Return	Air handling system Filter Schedule	Airflow rate
Airflow Path	Airflow element Filter Schedule Wind pressure option and Profile	Azimuth angle Constant wind pressure value Multiplier Relative elevation Wind speed modifier
Control Node *	Control type	N/A
Duct Junction	Color Schedule	Relative elevation Temperature
Duct Segment	Color Duct flow element Filter Schedule	Length Loss coefficient
Duct Terminal	Balanced Color Filter Schedule Wind pressure option and Profile	Azimuth angle Constant wind pressure value Balance loss coefficient Design flow rate Free face area Relative elevation Temperature Terminal loss coefficient Wind speed modifier
Occupant	Inhalation schedule	Body weight Multiplier Peak inhalation rate
Source/Sink	Source/sink element Schedule Species	Multiplier
Zone	1D zone Color Kinetic reaction Name Schedule	Floor area Temperature Volume

* Control nodes are not replaceable using this feature

Table – Find and Replace options

3.1.6 Viewing Results on the SketchPad

ContamW provides some graphical displays of simulation results upon the SketchPad. You can display airflow rates and pressure differences across airflow paths on the SketchPad if a simulation has been performed and results of the simulation are available (see Viewing Results in the Working with Results section of the manual).

3.1.7 Viewing Envelope Wind Pressure Differentials

ContamW provides a feature that allows you to visualize the effects of wind upon a building envelope. A graphical display of wind pressures at each of the flow paths that are adjacent to the ambient zone can be viewed on the SketchPad (see Checking Wind Pressure Data in the Defining Steady State Weather and Wind section). This feature enables you to verify input of wind speed and direction data with respect to the orientation of the building envelope to the wind.

3.1.8 Working with Project Files

All information related to the layout, building components, and occupant information of your CONTAM project are stored in a "project" file. When you save the project file, it will be stored with the ".PRJ" extension. CONTAM project file names must conform to the file naming conventions of the Windows operating system. This project name is displayed in the Title bar of the ContamW application window.

All project file operations are contained under the **File** menu. These operations include: **New Project**, **Open Project…**, **Save Project**, and **Save Project As…** Note that some of these file operations have keyboard short-cuts and/or tool bar buttons associated with them.

Creating a New Project

Menu Command: **File → New Project**
Keyboard Shortcut: None
Toolbar Button:

Use this command if you are currently working on a project and you wish to create a new one. This command will clear the SketchPad and all data related to the current project from ContamW's program memory. If your current project has not been saved prior to executing this command, you will be asked whether or not you wish to save it prior to clearing it.

A dialog box will appear enabling you to enter a project description (See **Editing the Project File Description** below)

Opening an Existing Project

Menu Command: **File → Open Project…**
Keyboard Shortcut: **Ctrl+O**
Toolbar Button:

OR

Menu Command: **File → Recent Projects**

Use these menu items to open and existing CONTAM project file. This command will display the File Open dialog box typical of Windows applications. This dialog box is set to display only

those files having the ".PRJ" extension in order to simplify your search for CONTAM project files. Follow the typical procedure for opening a file using the Windows operating system.

The Open dialog box provides you with both the Version and Description of the highlighted project file. You can edit the project file description via the **File → Project Description…** menu item.

When a project file is successfully read by ContamW, a backup copy of the project file is created with the name CONTAM.BKP in the program directory.

Opening Project Support Files

You can now open input and output files related to the current project, including the ContamW and ContamX log files, via the **File → Open Project Input Files** and **File → Open Project Output Files** menu commands. The files will open according to the Windows file extension associations.

Converting Projects from Previous Versions

Menu Command: **File → Open Project…**
Keyboard Shortcut: **Ctrl+O**

This is the same as opening an existing project. However, after you open a project of an earlier version it will be untitled, so you must give the project a name when saving it. Once you save a project with the most recent format, you will no longer be able to open it using an earlier version of CONTAM.

A utility is also provided on the NIST website that will enable you to quickly update a set of project files to the current version. For example PrjUp24.exe will upgrade to the CONTAM 2.4 format. To utilize this utility, copy prjup24.exe and the desired project file(s) to a new directory, select the projects to update and drag and drop them onto the prjup24.exe file. This will replace the existing files with updated versions.

Editing the Project File Description

Menu Command: **File → Project Description…**

Use this feature to enter a project file description which is provided to help you manage your project files. This description will be available when you are opening project files from within ContamW.

Saving a Project

Menu Command: **File → Save Project**
Keyboard Shortcut: **Ctrl+S**
Toolbar Button:

This option saves the building description to the project file under its current name.

Renaming a Project

Menu Command: **File → Save Project As…**
Keyboard Shortcut: None
Toolbar Button: None

Use this command to save a copy of the current project file under a new name. This will display the Windows "Save As" dialog box and allow you to specify both a storage location and a file name for the project file. ContamW will automatically append the ".prj" extension to the

filename, so you do not have to give the file an extension. If you do give the file an extension, ".prj" will be appended as well as the extension you provide. If you specify the name of an existing file, you will be warned that saving the file will overwrite the existing file. If you try to save an untitled or new project file prior to giving it a name, the "Save As" dialog box will be displayed.

Exiting ContamW

Menu Command: **File → Exit**
Keyboard Shortcut: None
Toolbar Button: None

You can also exit the program using the standard window-closing button located in the upper right-hand corner of the program window.

3.2 CONFIGURING CONTAMW

The Program and Project options of ContamW are available from the **View** menu. Program options can be saved to the CONTAM.CFG file located in the CONTAM Program Directory. Upon startup, ContamW looks for this file and loads the saved settings if the file exists. This way you can set ContamW to start with the same default settings each time you run the program. Project options are saved with the current project file.

NOTE: The path name of the EWC-to-WPC file converter does not appear in the configuration dialog boxes; however, if a converter is selected when you click the Save button it will be saved to the CONTAM.CFG file. The value saved will be that which you have set in the **WPC File…** selection of the **Weather** menu. To reset the converter name back to "null," you must edit the configuration file manually.

3.2.1 Project Configuration Properties

All of these settings are saved within the CONTAM project file.

Units

The following parameters define the units and temperature that will be used by default throughout the project when you are providing input to the program. You do not have to use these units throughout the project; however, you should select the primary set of units for convenience when entering data throughout a project. You can change these defaults at any time while working with a project, and you can select different units for individual parameters as you enter them.

System of Units: You can select the appropriate radio button to set the system of units that you would like the program to use. You can either select the International System of Units (SI) or the Inch-Pound (IP) system of units.

Units of Flow: You can select the appropriate radio button for the most commonly used units of flow in the current project. These are the units in which airflow simulation results will be displayed within the status bar. ContamW converts all airflow rates to and performs all calculations in mass flow units of kg/s. Note that all volumetric airflow rate units are at standard temperature and pressure.

Default Zone and Junction Temperature: This value will be the default temperature used whenever you create a new zone, duct junction or terminal within the current project. If you change this value, you will be prompted as to whether or not you would like to update the temperatures of all existing zones, junctions and terminals.

SketchPad Dimensions

Size: This is the width and height of the SketchPad. These dimensions are presented in both cell units and units of length as defined by the Scaling Factor below. The SketchPad is made up of square cells that represent each space wherein an icon may be placed. The default (and minimum) values are 66 cells wide and 58 cells high.

Scaling Factor: Sets the dimension of a cell in order to calculate SketchPad locations and wall dimensions in the units provided when displaying pseudo-geometry on the SketchPad.

Show Pseudo-Geometry: The pseudo-geometry drawing mode uses the Scaling Factor to present location and wall dimensions on the SketchPad in the Status Bar. This feature can be useful when working with the TRNSYS-CONTAM coupling capability that requires geometrical data for the TRNSYS building representation used by TRNSYS Type56. Pseudo-geometry can also be useful when setting airflow path multipliers because wall areas can be displayed. The extents of items (dx and dy) will be displayed in the Status Bar as you draw them.

Origin: Sets the location (column and row) of the origin to be used when calculating the position of items on the SketchPad. A special icon will appear on the SketchPad at the specified location. By default the origin represents the bottom-left of the SketchPad grid (column 1, row 56) and y-values are positive above the origin. You can invert the y-axis so that y-values are negative above the origin.

NOTE: You can also set the origin by right-clicking on the SketchPad and selecting "Set Origin" from the context menu.

3.2.2 Program Configuration Properties

Use the Save button at the bottom of this dialog box to save the CONTAM.CFG file.

Units

The Program units define the units and temperature that will used to populate the Project units (presented above) when you create a new project file.

SketchPad Display

Show Floating Status: Check this box to display the Floating Status box on the SketchPad. You can also turn the floating status box on and off via the View → Floating Status (Ctrl+T) menu command.

Show Zone Colors: Check this box to display zone fill colors on the SketchPad. If this box is not checked, then all zones will be filled using the "Background" palette color. You can also turn the color zones option on and off via the **View → Color Zones (Ctrl+Z)** menu command.

Show Duct Colors: Check this box to display duct colors on the SketchPad. If this box is not checked, then all ducts will be displayed using the "Ducts" palette color. You can also turn the color ducts option on and off via the **View → ColorZones (Ctrl+Z)** menu command.

Maximize ContamW at startup: Check this box to maximize the ContamW window when the you start the program.

Cell/Icon Size

Current Cell/Icon Size: There are seven options for displaying icons on the SketchPad (1x1, 2x2, 3x3, 4x4, 5x5, 8x8, and 16x16). The default option is 8x8. This is a "zooming" feature that provides a visual aid to enable you to view sketches in more detail or to fit larger sketches within the program window. The cell size has no effect on the project simulation. There are two "zooming" buttons provided on the toolbar to quickly increase and decrease the cell size according to the available cell sizes below.

Available Cell Sizes: Use these check boxes to select which sizes you want available. The 1x1 size causes problems with some versions of the Window 98 operating system. If this is the case for you, then deselect the 1x1 check box and click the "Save Configuration" button.

Cell Jump: Use this to set the number of SketchPad cells the cursor will jump when you use the **Shift + Arrow Keys** to maneuver around the SketchPad.

Colors

Palettes: ContamW provides the ability to modify the color of items displayed on the SketchPad. You can select one of four available palettes. You can not modify the colors of the "Default" palette, but you can modify the other three "User Palettes." Select a palette to display the colors of the palette in the **Palette Colors** section below.

Palette Colors: These are the colors use to display items on the SketchPad. Click the color box to change the color used to display each type of item. Only the colors of the User Palettes can be changed. The following is a brief description of each category.

Foreground - Defined icons including walls

Background - Background of SketchPad cells, default zone fill and fill of all zones when **Color Zones** option is not active.

Flow Results - Lines displaying scaled flow results for airflow paths and duct terminals

Pres Results - Lines displaying scaled pressure difference results across airflow paths in the Results and Wind Pressure view modes

Drawing Line - Cell highlight when drawing on the SketchPad

Undefined - Undefined icons

Controls - Defined control icons

Ducts - Default color of defined ducts and color of all defined ducts when **Color Ducts** view option is not active

Errors - Background color of cells when an error condition is being highlighted on the SketchPad. Errors are indicated when a building check fails. Building checks are performed when you choose **Run Simulation** or **Create ContamX Input File** from the **Simulation** menu.

Cursor - SketchPad drawing and icon selection cursor

Highlight - Icon highlighting color for find operation

Sub Level - All icons on sub-level when sub-level display option is activated via the **Level → Reveal Level Below (Ctrl+Shift+B)** menu command.

Results Display

Results Lines: You can select whether or not to display the airflow rates and pressure differences across airflow paths on the SketchPad when in the *Results Display Mode*.

Both Pressure and Flow: Display both the airflow rates (green lines) and pressure differences (red lines).

Pressure Only: Display only the pressure difference lines.

Flow Only: Display only the airflow lines.

Time Step Navigation: Set the number of simulation time steps to skip when using the Results Navigation jump commands.

See Viewing Results in the Working with Results section.

3.2.3 Working with Walls

In ContamW, walls are used to create enclosed regions referred to as zones on a given level. Walls include the building envelope and internal partitions with a significant resistance to airflow. You will note when drawing on the SketchPad that you can only draw walls in odd numbered rows and columns. This is done to avoid conflicts with ducts that can only be drawn on the even rows and columns of the SketchPad.

It is important to realize that ContamW is not meant to provide drawing capabilities similar to that of a Computer Aided Drawing (CAD) program. The SketchPad drawing functions were designed to provide a strict environment for creating the input to the underlying multizone model upon which the CONTAM simulation engine is based.

3.2.4 Drawing Walls

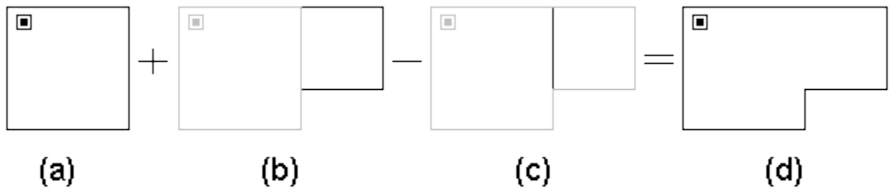
You can draw walls using either of the two wall drawing tools previously described in the "Working with the SketchPad" section. Use the *box drawing tool* to quickly draw a rectangular region and the *wall drawing tool* to draw a free form wall. Use the wall drawing tool to draw almost any shape wall. Walls must always form complete enclosures. Therefore, a wall cannot have an open or dangling end. It also may not be drawn across building component icons. You will receive a warning message if you attempt to draw an invalid wall.

3.2.5 Deleting Walls

You can delete a wall by moving the caret to any portion of the wall and selecting **Delete** from the **Edit** menu or using the keyboard shortcut **Del** key. ContamW will highlight the section of wall to be deleted and request confirmation to delete the indicated section. If the caret is on the intersection of three or more walls, you will be given multiple options of wall segments to delete. If you select "No" when asked to confirm deletion, the next option for deletion will become highlighted until you either delete a section or all of the options are exhausted.

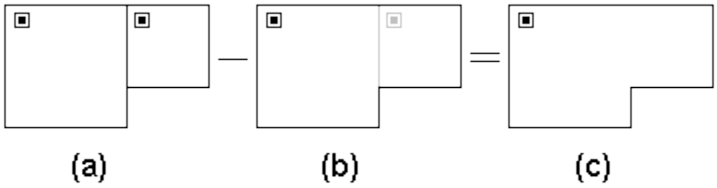
3.2.6 Modifying Walls

You modify the positions or shapes of walls by adding and deleting wall sections. For example in the following figures, if you want to modify the zone in figure (a) to obtain the zone in figure (d), you would first add the dark line in figure (b) then delete the dark line in figure (c).



When modifying walls you may need to move or delete other icons. For example in the following figures, to create drawing (c) from drawing (a), you would delete the lighter line and

the lighter zone icon from figure (b). You must delete one of the two zone icons, or you will end up with two zone icons within the same enclosed wall area. This is not permitted, and you will receive a message indicating a zone definition error.



3.3 WORKING WITH LEVELS

CONTAM projects are organized by levels. Each level of a CONTAM project contains a plan view drawing. Typically, each level of a CONTAM project corresponds to a floor of a building. Depending on the detail required for your modeling purposes, you could represent a building floor using multiple levels. For example, a level could be used to represent a ceiling or floor plenum.

3.3.1 Creating Levels

When you first start ContamW, a default level is created so that you can begin working on a drawing right away without having to create a new level. There are three commands in the **Level** menu that you use when creating new levels. These are the **Copy Level**, **Paste Level**, and **Insert Blank Level** commands. Whenever you create a new level, whether it is blank or a copy of another level, ContamW will give it a default name that will consist of a number enclosed within the "<" and ">" characters. You can modify this name later by editing the data associated with a level.

You must be careful when copying levels within ContamW. For example, you might have an air handler defined within a duct system on a level and then copy it to another level. This would create another air handler on the new level. If the ductwork is connected between the two levels, the two air handlers may act against each other. You must be careful to make connections between building levels in a manner that makes sense for your purposes.

Creating Blank Levels

Menu Command: **Level** → **Insert Blank Level** (**Above**/**Below Current Level**)
Keyboard Shortcut: None
Toolbar Button: None

Use these commands to create a blank level. When you create a blank level, you must select whether you want it created above or below the current level.

Copying and Pasting levels

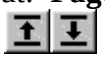
Menu Command: **Level** → **Copy Level** along with
Level → **Paste Level** (**Above**/**Below Current Level**)
Keyboard Shortcut: None
Toolbar Button: None

With ContamW you can copy an entire level and insert it as an entire level anywhere within the current project. Use the Copy Level command to copy the level currently displayed on the SketchPad; move to either the level above or below where you wish to insert the copied level and use one of the Paste Level commands to insert the copied level.

3.3.2 Viewing Levels

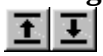
If a project has more than one level, you can specify the level that you want to view or modify on the SketchPad by using the level commands. All level-related commands are located under the **Level** menu. There are also some shortcut keys and toolbar buttons provided for your convenience.

Moving Up/Down a Level

Menu Command: **Level → Go to (Level Above/Below)**
Keyboard Shortcut: **Page Up**, **Page Down**
Toolbar Button: 

Use this command to change which level you want the SketchPad to display.

Changing the Currently Active Level

Menu Command: **Level → Go to (Level Above/Below)**
Keyboard Shortcut: **Page Up**, **Page Down**
Toolbar Button: 

Use this command to change which level you want the SketchPad to display.

Displaying Multiple Levels

Menu Command: **Level → Reveal Level Below**
Keyboard Shortcut: **Ctrl+Shift+B**
Toolbar Button: None

Use this command to see both the current level and the level below the current level at the same time. ContamW will display the walls and building component icons of the level below in gray. This feature is useful for aligning building features between adjacent levels.

3.3.3 Deleting Levels

Menu Command: **Level → Delete Level**
Keyboard Shortcut: None
Toolbar Button: None

Use this command to delete an entire level. Once you have deleted a level, you cannot undo the deletion. You may want to save a copy of the file prior to deleting a level; this is the only way to prevent losing your work.

3.3.4 Modifying Levels

To modify level data, you access the Level Data dialog box using the **Edit Level Data…** command of the **Level** menu. You can also use the keyboard shortcut **F8** to display the dialog box. The following section shows the information contained on the "Level Data" dialog box.

3.3.5 Level Data

This is the information associated with each level that you create. ContamW will provide default values for this data, but you can modify it as required for your particular building.

Name: Name to identify the level. All level names must be unique.

Elevation of this level: The elevation of the base of the level above ground.

Distance to level above: This is the height of the level from floor to ceiling. ContamW will use this value to calculate zone volumes based on the floor area of each zone. If you change this value, a dialog box will appear asking if you would like ContamW to adjust the volumes of the zones on the level based on the new level height. If you select Yes, then the zone volumes will

be adjusted by $V_{new} = V_{old} * H_{new} / H_{old}$. If there are *phantom zones* on the level, you will be advised that their volumes were not adjusted, so you should handle those zones individually.

3.3.6 Checking Levels

Menu Command: **Level → Check Current Level**
Keyboard Shortcut: None
Toolbar Button: None

Use this feature to check the data for the currently displayed level and make sure the building components are defined. Errors will be highlighted in red on the SketchPad.

3.4 WORKING WITH ZONES

A *zone* is a volume of air separated from other volumes of air by walls, floor and ceiling. Generally, zones are characterized by having a uniform temperature and pressure (that varies hydrostatically) and contaminant concentration. However, the concentration within user-selected zones can be allowed to vary in one dimension (See 1D Zone Data Properties within this section) via the 1D convection/diffusion model (See the Theoretical Background section). In addition, there are three different types of zone icons: *normal*, *phantom* and *ambient*. A *normal* zone icon represents a zone that is separated by a floor from the zone below. A *phantom* zone is used to indicate that there is no floor below the enclosed region on a given level. That is, the region on this level is actually part of the zone below and has the same temperature and contaminant concentration. An *ambient* zone icon represents the outdoor air. The ambient zone, which surrounds the building, is already identified by default by the icon at the upper-left corner of the SketchPad.

You create zones on the SketchPad by first drawing walls that enclose a region, placing a single zone icon within the enclosed region, and then defining the zone icon. Eventually, every zone must be connected to either the ambient zone or a constant pressure zone either directly or indirectly via any set of paths that prevents the zone from being *isolated* from a zone of "known" pressure.

3.4.1 Creating Zones

You create zones by drawing walls upon the SketchPad. The wall drawing operations are described in the Drawing Walls section of this manual. The shape and size of zones as drawn upon the SketchPad do not provide the underlying model with any scaling information. However, the manner in which the enclosed regions border each other is significant. Any zones between which you wish to provide a direct connection (via an airflow path) must share a common wall. Therefore, when drawing a building floor plan, you should try to maintain the general topology of the actual floor plan. While scale is not significant, ContamW does provide you with the SketchPad coordinates that may help you when laying out your project.

The dimensions of your zones are determined when you define them. If you change the shape or size of a region on the SketchPad that contains a zone icon, the dimensions do not change according to the model unless you actually modify the properties of the associated zone icon.

3.4.2 Deleting Zones

You delete a zone by first deleting the zone icon. After you delete the zone icon, delete the wall or walls necessary to eliminate the enclosed region that is left behind. You must avoid the existence of an enclosed region without a zone icon; CONTAM will not perform a simulation with an undefined zone icon in a project.

3.4.3 Modifying Zones

You modify the SketchPad representation of a zone by adding and removing walls. This procedure is described in the Modifying Walls section of this manual. To modify the parameters of a zone, you use the icon definition procedure (See Defining Building Component Icons) to display its properties and make the desired changes.

Once you define a zone, you can move the zone icon anywhere within the zone using the **Cut** and **Paste** functions of the **Edit** menu. These commands will only allow you to paste the icon into the zone in which it is currently located.

3.4.4 Defining Zones

After you draw the enclosed region of a zone, you must define the zone. To define a zone you must first draw a zone icon within the enclosed region (See Drawing Building Component Icons) and then use the icon definition procedure (See Defining Building Component Icons) to display and edit the properties of the zone.

The properties of each zone include a name, temperature (constant or scheduled), pressure (variable or constant), volume, information describing contaminant behavior within the zone, and 1D zone information. You must provide each zone with a name that is unique to each level of a project. For this reason, you cannot copy zone data within a level. However, you can copy entire levels of data, including zone data, from one level to another (See Creating Levels). Detailed descriptions of zone properties are given under Zone Properties below.

Only *normal zones* require definition. *Ambient* and *phantom* zone icons appear as defined icons as soon as you place them onto the SketchPad. However, you must include the volume of *phantom zones* in the volume of the associated *normal zone* below it.

3.4.5 Zone Properties

This section provides detailed descriptions of the specific zone properties. The following sections are the context-sensitive help topics that you can access by pressing **F1** when working with property pages of the "Zone Properties" property sheet.

Zone – Zone Data Properties

These are the basic properties that describe a zone.

Zone Name: This is the symbolic name of the zone. Enter a name up to 15 characters in length. Zone names must be unique within each level of a building.

Dimensions

> **Volume:** Zone volume is used in the dynamic contaminant calculations. For phantom zones, the zone volume is set at the standard zone icon on the lowest level for the entire height of the zone and includes the volume of the phantom zones on the levels above.
>
> **Floor Area:** Instead of the volume, you may enter a floor area. Floor area is then multiplied by the height of the current building level to compute a volume. This area is not used for any other purpose by CONTAM.
>
> **Color:** Select a zone fill color to help you distinguish zones on the SketchPad. You must activate the **Color Zones** option from the **View** menu in order to display zone colors on the SketchPad.
>
> **Include in Building Volume:** This refers to the calculation of whole building *air change rates* that you can now have CONTAM perform during simulation. The associated check box is checked by default, so if you do not want the zone to be included in the whole *building volume* you should uncheck this box. The *air change rate* and *building pressurization* calculations will calculate the total rate of airflow from *ambient* into all of the zones having

this box checked. Typically, you would include all *conditioned space* in the building volume; for example, you might not want to include an attic or crawl space.

Temperature & Pressure

Temperature: Set the value you want to use as the constant temperature of the zone. Whenever you create a new zone, ContamW will use the default value, which you can override by entering another value. You can set the default zone temperature via the **Options…** command of the **View** menu. Optionally, you can define a Schedule for the zone temperature via the Schedule properties (See Schedule Properties for zone temperature).

Pressure: The zone air pressure relative to ambient can be specified as either variable or constant. Typically you would set the zone pressure to be variable and allow it to be calculated during the simulation. However, you might set the pressure to be constant if you want to simulate a *fan pressurization* (*blower door*) test of a building (see Building Pressurization Test in the Applications section of the manual) or to perform analytic test cases of CONTAM.

Zone – Contaminant Properties

These are the contaminant-related properties of a zone. Contaminants must be defined prior to defining contaminant data for a zone. To define contaminants, select **Data** then **Contaminants…** from the main program menu.

Zone Name: This is the symbolic name of the zone as entered on the Zone Data property page.

Contaminant Concentrations: Select *variable* or *constant*. Typically you would set this to *variable* and allow ContamX to calculate the contaminant concentration within the zone during simulation. However, you may wish to set this to *constant* as a simple means of creating a simple contaminant source within a zone. If you set this to *constant*, the contaminant concentrations in this zone will begin and remain at the values you set for the initial concentrations.

Initial Concentration: Select a contaminant from the list and enter the *initial contaminant concentrations* for dynamic (transient) simulations. Note that only those species you've selected to be contaminants (to use during simulation) appear in the list. If a species for which you wish to set an initial concentration does not appear in the list you must set "Use in simulation" property of the corresponding species to be true (See Creating Species and Contaminants). You can reset these initial values through the Run Control Properties of the Simulation Parameters.

IMPORTANT: Changing the number of contaminants, i.e. those species used in the simulation, will reset the *initial concentrations* of all zones to the default contaminant concentrations. You can reset all of these concentrations via the Run Control Properties as indicated above or individually here.

Sum of Non-trace Initial Concentrations: This value is provided as a check for you to insure that you input a reasonable set of initial values when using *non-trace contaminants*. This value should be very close to 1.0.

Kinetic Reactions: If there are previously defined kinetic reactions within the current project, you may select one from the list of names. Click the "New Reaction" button to define a new kinetic reaction. To view or modify existing kinetic reaction data click the "Edit Reaction" button. Setting this field to <none> indicates no reactions in the zone. You can also import kinetic reactions from a contaminant-related library file (See Working with Data and Libraries).

Zone – Detailed Zone Properties

These are the properties used to define 1D convection/diffusion zones, in which contaminant concentrations can vary along a user-specified axis. This option is provided for cases in which you need to account for the delay in contaminant transport within long or tall zones. If this feature is enabled, ContamX will subdivide this zone along the specified axis into multiple cells or sub-nodes for the purposes of contaminant transport calculations.

Zone Name: This is the symbolic name of the zone as entered on the Zone Data property page.

Zone Type

- Normal Zone: well-mixed zone
- 1D Zone: 1D convection/diffusion zone
- CFD Zone: Only one CFD zone is allowed per CONTAM project. Use the CFD0 Editor (available via the NIST IAQanalysis website) to define the CFD properties of the zone, e.g., zone mesh and boundaries.

1D Zone Data

Axial Dispersion Coef: Enter the axial dispersion coefficient and units for the zone. Note that the zone dispersion coefficient is zone specific and is not necessarily the molecular diffusion properties of the defined contaminants. This coefficient is empirically based and should account for turbulent mixing as well.

Axis End Point Coordinates: Define the end points, **Point 1** and **Point 2**, of the axis of the 1D zone. These coordinates will define the direction along which contaminant transport will be calculated when treating this zone as a 1D convection diffusion zone. These values are required to establish the relationship of other building components with respect to this zone, i.e., flow paths, duct junctions and terminals, simple air-handling unit supply and return points, contaminant source/sinks, sensors and occupants.

While the coordinate system is user-defined, ContamW will verify the relationship of 1D zones to associated building components prior to running a simulation to insure that the coordinates "match up." This verification is performed automatically when you select either the **Run Simulation** or **Create a ContamX Input File** command from the **Simulation** menu. If there are discrepancies, messages will be displayed notifying you of the nature of the discrepancies.

X and Y: The X and Y values are in absolute coordinates. These values can be based on scale drawings and/or a user-defined coordinate system. Unlike walls on the SketchPad, axes are not required to be rectilinear, i.e., you may specify axes that are not horizontal or vertical in the x-y plane.

Rel Elevation: Enter the Z-coordinate as the height relative to the base of the zone.

Cell Size: This is the cell size into which ContamX will sub-divide the zone during simulation if the zone is to be treated as a 1D convection/diffusion zone. If the axis length is not evenly divisible by the cell size, then the cell size will be adjusted as needed. While there is no strict guidance on the selection of cell size, it should generally be on the order of the flow velocity along the axis times the simulation time step.

Units: Set the units of the above items.

Axis Orientation on the SketchPad: The orientation of the axis on the SketchPad is useful when using a results viewing tool such as ContamRV. ContamRV can display the variation of concentration within 1D zones if the simulation output options are set to write to the 1D results files (See Simulation - Output Properties in the Simulation Parameters section). Mass fractions of 1D zone cells are written to the 1D results file in the order from Point 1 to Point 2 of the Axis End Points. The "Axis Orientation" enables ContamRV to color the rectilinear region of the zone on the SketchPad even if the axis endpoints specify an axis that is not actually horizontal or vertical in the plan view, e.g., a slanted hallway.

CFD Zone Data

CFD Zone ID: This identifier is used to name the CFD files associated with this project. These files will be generated by the CFD0 Editor and ContamX. For example, using the CFD0 Editor you import a PRJ file that contains a CFD Zone. Once you have used the CFD0 Editor to define the CFD domain, you will use the CFD0 Editor to generate a CFD file having the same name as the PRJ file but with the CFD Zone ID appended to it, e.g., MyProject_CfdZone.cfd. Running a coupled simulation will generate more similarly named files having various extensions.

Zone – Schedule Properties

These are the properties used to define a temperature schedule for the zone. If a temperature schedule is associated with the zone then the constant value will be overridden and the simulation run control should be set to vary the density within building zones during simulation (See Airflow Numerics Properties in the Working with Simulations section).

Zone Name: This is the symbolic name of the zone as entered on the Zone Data property page.

Temperature Schedule: Select the temperature schedule you want to associate with this zone. (See Working with Schedules).

CVF/DVF Control: Values from either a CVF or DVF file can be used in place of the above Schedule (read about CVF and DVF controls in the Control Element Types section). This enables using these control element types without having to draw/define an associated control network. The associated radio buttons "Use Neither", "Use CVF" and "Use DVF" are used to select the desired control method. Select the heading name from the associated "Name" drop down box that will display the headings of either the CVF or DVF file as per the aforementioned radio button selection.

Control Status: This will display the current method of control: Schedule, direct CVF, direct DVF or Control Network. If an actuator of a control network on the SketchPad is associated with the scheduled item then it will take precedence over the Schedule and CVF/DVF Control. If either the "Use CVF" or "Use DVF" radio button is selected then CVF or DVF will take precedence over an associated schedule.

3.5 WORKING WITH AIRFLOW PATHS

An airflow path is a CONTAM building component through which air can move between two adjacent zones. These components can be cracks in the building envelope, open doorways, exhaust fans, etc. The air pressures in the adjacent zones and the flow characteristics of the flow path itself determine the flow through most of these paths. The location of the SketchPad icon that represents a flow path determines the two zones that are connected by a flow path. If the path icon is placed on a wall, it connects the zones on opposite sides of the wall. If it is placed in a blank cell of a zone (i.e. on the floor), it connects the zone containing the flow path icon to the zone located directly below the flow path icon.

You can place airflow paths on the SketchPad, define their flow characteristics, move, copy, and delete them. You must provide each airflow path with specific information that describes its flow characteristics. Much of the information that describes the flow characteristics of airflow paths is contained in *airflow elements* that you must create when defining the airflow paths. The details of drawing, defining and modifying airflow paths are described in the following sections.

3.5.1 Airflow Elements

Each airflow path must refer to an airflow element. *Airflow elements* describe the mathematical relationship between the flow through an airflow path and the pressure drop across the path. ContamW provides you with several mathematical models or types to choose from. Each of these airflow element types is described in detail in the Airflow Path Properties section. While every airflow path must refer to a single *airflow element*, multiple paths can refer to the same *airflow element*. *Airflow elements* can also be stored within a CONTAM library file and shared between different CONTAM project files (See Working with Data and Libraries).

3.5.2 Creating Airflow Paths

You create airflow paths by placing an airflow path icon on the SketchPad (See Drawing Building Component Icons). Because flow paths connect only two adjacent zones, there are several restrictions on the placement of flow paths. You may not place airflow path icons on the corner of a zone or on the blank cell of a level that does not have a zone located directly below it. Note that you can place an airflow path icon on the "roof" of a building that connects a zone below to the ambient. In this case, the roof level would be a level that does not contain walls but contains airflow path icons located in the floor of the roof level (above zones on the level below). The context-sensitive feature of the pop-up icon placement menu enforces these restrictions. The "Flow Path" menu selection will be disabled (grayed out) if you pop-up on a location where ContamW does not allow the icon to be placed.

This list shows the various flow path icons and provides a brief description of each.

Icons	Description
	Small and large one-way flow paths
	Small and large one-way flow paths
	Directional fan flow paths

3.5.3 Deleting Airflow Paths

You delete airflow path icons using the icon deletion procedure (See Deleting Building Component Icons). Deleting airflow path icons from the SketchPad does not delete any airflow elements that you may have defined. These flow elements can only be deleted using the CONTAM Library Manager, accessible through the **Data** menu for each type of data element.

3.5.4 Modifying Airflow Paths

To modify the parameters of an airflow path, you use the icon definition procedure (See Defining Building Component Icons) to display its properties and make the desired changes. You can also move and copy airflow paths.

Moving Airflow Path Icons

Once you define an airflow path, you can move the path icon using the **Cut** and **Paste** functions of the **Edit** menu. ContamW will constrain the movement of path icons according to the following rules. You can only move a flow path through a wall onto a wall, not onto the floor of a zone. You can only move a flow path through a floor to another location on the floor, not onto a wall. You can only move a path icon within the same level upon which it is located.

Copying Airflow Path Icons

Once you define an airflow path, you can copy the path icon using the **Copy** and **Paste** functions of the **Edit** menu. ContamW will constrain the copying of path icons according to the following rules. You can only copy a flow path through a wall onto a wall, not onto the floor of a zone. You can only copy a flow path through a floor to another location on the floor, not onto a wall. You may copy a path icon from one level to another.

3.5.5 Defining Airflow Paths

After you draw a *Flow Path icon* on the SketchPad you must define it using the icon definition procedure (See Defining Building Component Icons) to display and edit the properties of the airflow path. This will display the "Airflow Path Properties" property sheet. Detailed descriptions of all airflow path properties are given in the Airflow Path Properties section of this manual. Once you have defined the properties of an airflow path, the icon will be displayed in black. The icon representing the defined airflow path may be slightly different from the original undefined icon depending on the type of airflow element you associate with the path (See the list below). From now on, you use this icon to access the properties of the airflow path.

Some of the properties associated with an airflow path are optional: *contaminant filtration*, *schedule*, *wind pressure* and *pressure limit*. You can associate contaminant filters with any flow path, and you can define a filter for each contaminant contained in your CONTAM project. A filter could be used, for example, to simulate the penetration of particles through a wall cavity. You can use a schedule to vary the flow through a path as a function of time. For example, you could use a schedule to open and close a path that represents a window. CONTAM allows you to account for the direct effects of wind pressure on the envelope of a building. The envelope is considered to be any wall that is adjacent to the ambient zone. If an airflow path is directly connected to the ambient zone, a "Wind Pressure" property page will be displayed to allow you to account for the effects of wind pressure on the path through the building envelope. Pressure limits are useful when you are using CONTAM to design a *smoke control system*. For example, you could set pressure limits (maximum and minimum) for stairwell doors so that those

doorways for which the pressure limits are violated will be highlighted upon the SketchPad when you view simulation results.

Airflow Path Properties

This section provides detailed descriptions of the specific airflow path properties. Each of the following subsections are the context-sensitive help topics that you can access by pressing **F1** when working with property pages of the "Airflow Path Properties" property sheet.

Airflow Path – Airflow Element Properties

Path Number: This is the number that appears in the status bar when you highlight a path icon. ContamW automatically assigns this number to each *airflow path* once they are defined. This number could change as you add and remove paths to or from a project. Each time you save a project, the path numbers are reassigned. Numbers are assigned beginning on the top level, starting from the upper-left corner of the SketchPad, moving left-to-right and top-to-bottom of the SketchPad. Paths include supplies and returns of simple air handling systems.

Airflow Element Name: <required> You may select a previously defined airflow element from those contained in the Name list. Click the "New Element" button to define a new airflow element. To view or modify airflow element data click the "Edit Element" button. You can also access the CONTAM Library Manager to import airflow elements from library files by clicking the "Library" button (See Working with Data and Libraries). Every airflow path must have an airflow element associated with it. This field may not be left blank.

Model Summary: This summarizes the information associated with the airflow element whose name is currently displayed in the Name field.

Airflow Path – Flow Path Properties

Relative Elevation: The elevation of the path will determine how it responds to, and influences, the building stack effect. Enter the elevation of the flow path relative to the elevation of the building level on which it is located.

For most flow paths (those not associated with a two-way flow model) enter the elevation of the mid-height of the flow path. Enter the elevation of the bottom of the opening for flow paths associated with a two-way flow element (See Airflow Element Types).

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" flow element, complete the description here by giving the reference area, length, or number of units.

Positive Flow Direction: In the cases of fans and flow/pressure limits (see below) it is necessary to know which direction is defined as positive. The arrows indicate the positive direction for flow of fans (forced flow element types) and flow and pressure drop for flow/pressure limits.

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.

Location: These are the absolute X and Y coordinates and units for the selected airflow path. The Z coordinate will be taken as the *Relative Elevation* of the airflow path with respect to the building level on which the path is located.

These values are required for flow paths through the building envelope when using a *Wind Pressure and Contaminant (WPC)* file (see Wind Pressure and Ambient Contaminant Files) or for any path that connects to a *1D convection/diffusion zone* (see 1D Zone Data). These coordinates will be verified by ContamW against a WPC file or the axis of the 1D zones to which the path connects. This verification is performed automatically when you select either the **Run Simulation** or **Create a ContamX Input File** command from the **Simulation** menu.

Airflow Path – Schedule Properties

The schedule value operates proportionally on the multiplier of the airflow path.

Schedule: If there are previously defined schedules within the current project, you may select one from the Name list. Click the "New Schedule" button to define a new schedule. To view or modify an existing schedule click the "Edit Schedule" button. Setting this field to <none> indicates no schedule. (See Working with Schedules)

CVF/DVF Control: Values from either a CVF or DVF file can be used in place of the above Schedule (read about CVF and DVF controls in the Control Element Types section). This enables using these control element types without having to draw/define an associated control network. The associated radio buttons "Use Neither", "Use CVF" and "Use DVF" are used to select the desired control method. Select the heading name from the associated "Name" drop down box that will display the headings of either the CVF or DVF file as per the aforementioned radio button selection.

Control Status: This will display the current method of control: Schedule, direct CVF, direct DVF or Control Network. If an actuator of a control network on the SketchPad is associated with the scheduled item then it will take precedence over the Schedule and CVF/DVF Control. If either the "Use CVF" or "Use DVF" radio button is selected then CVF or DVF will take precedence over an associated schedule.

Airflow Path – Filter Properties

Filter: You may either create, edit or delete the filter associated with this flow path. Creating a new filter or modifying an existing one will display the Filter Properties dialog box. Use this dialog box to create and edit filter elements as needed. For an explanation of contaminant filters see Contaminant Filters in the Using CONTAMW section of the manual.

Airflow Path – CFD Properties

Airflow paths connecting CFD zones to non-CFD zones are referred to as *interface paths* whose properties are defined by the following parameters.

CFD Path ID: Each interface path must be given a unique identifier. This identifier will be used to identify each interface path within the *CFD0 Editor* that is used to define the size and location of the interface paths with respect to the CFD zone.

Boundary Condition - CONTAM to CFD: Select the type of boundary condition to pass from the multizone to the CFD calculation for this path. These options will be enabled/disabled depending on the multizone-to-CFD coupling method selected (see CFD Numerics) and the type of airflow element of the interface path. The "CONTAM <-> CFD" and the "CONTAM -> CFD->CONTAM" coupling methods will set all boundary conditions to their default values to provide the pressure-pressure boundary condition exchange method, which is the most stable of the three (pressure-pressure, pressure-flow, flow-pressure).

- **Mass Flow Rate:** CONTAM provides a mass flow rate boundary condition to CFD.
- **Pressure:** CONTAM provides a pressure boundary condition to CFD.

Pressure Boundary Condition Type

Linear: When a pressure boundary condition is provided by CONTAM to CFD, the linear coefficient (C = 1.0) and exponent (Exp = 1.0) will be used for each interface cell in the CFD zone. This simple model should be used for large openings with potential two-way flow (recirculation) across the openings, where the "Stagnation and Static Pressure Model" normally cause numerical issues. This is the default model.

Stagnation and Static Pressure: When a pressure boundary condition is provided by CONTAM to CFD, a stagnation pressure is imposed for inflows of the CFD zone, and a static pressure is imposed for outflows of the CFD zone. Numerically, a static pressure is set for CFD outflows as follows: a linear coefficient with the order of 10^3 is used so that the CFD local pressure can be maintained as a difference of 10^{-3} from the external boundary pressure. This may cause numerical issues for large openings with two-way flows, where a slight change of dynamic pressure head could cause a huge airflow at the boundary. In such cases, a linear model should be used. When comparing a coupled simulation to CFD-only simulation (all zones simulated by CFD), the flow coefficients and exponents in a coupled simulation are very important, which are normally unknown. Above all, a linear model brings more numerical stability while a stagnation and static pressure model is a means of setting CFD pressure boundary conditions.

Airflow Path – Wind Properties

Wind Pressure Option: There are three wind pressure options for openings in the building envelope:

1. No wind pressure
2. Constant pressure
3. Pressure dependent on wind speed and direction

When you select one of the above options, the following data entry options that correspond to the wind pressure option that you selected will be made available for your input.

Constant Pressure Data: Enter a constant wind pressure (wind speed and direction independent).

Variable Pressure Data: Variable wind pressures are computed from the product of the following three parameters:

1. The dynamic pressure of the wind at some reference height. ContamX will determine this pressure based on either the steady state wind data (see Wind Properties) or the transient wind data from a weather file (see Defining Transient Weather and Wind).
2. A coefficient accounting for local terrain effects (Wind Pressure Modifier).
3. A pressure coefficient accounting for relative wind direction (Wind Pressure Profile).

Wind Pressure Modifier: CONTAM uses this value to account for differences between wind velocity profiles at the building site and that at a measurement location, e.g. an airport weather station (See Working with Weather and Wind). CONTAM only requires this value if you are implementing Variable Wind Pressure for an airflow path. CONTAMW will provide a default value for this based on the data entered via the Weather and Wind Parameters Property Page or you can override the default value for a particular airflow path.

Wall Azimuth Angle: Enter the direction the wall faces (clockwise from north).

Profile: The wind pressure profile accounts for the wind direction effects. You may select a previously defined wind pressure profile from those contained in the Name list. Click the "New Profile" button to define a new wind pressure profile. To view or modify wind pressure profile data, click the "Edit Profile" button. After you press the "New Profile" or "Edit Profile" button, the Wind Pressure Profile page will be displayed with a graphical representation of the profile.

Wind Pressure Profile

Wind Pressure Profiles are used to account for the wind direction effects in flow elements. CONTAM refers to the function relating the average wind pressure coefficient for the face of a building to the angle of incidence of the wind on the face of the building as the *wind pressure profile* or $f(\theta)$. A more detailed explanation of this function is given in the Working with Weather and Wind section. The profile is displayed graphically at the bottom of the screen for a visual review of the data.

Name: This is the name you give to the wind pressure profile. Wind profile names must be unique within a project.

Description: Field for entering a more detailed description of the specific wind pressure profile.

Data Points: You may enter up to 16 angle/pressure coefficient pairs. Pressure coefficients are in the range -1 to 1. The first value must be at a wind direction of zero degrees. The angle zero indicates a wind blowing directly toward the surface in which the opening lies. An identical value at angle 360º is automatically assumed. 90º indicates a wind blowing parallel to the wall from the right, and 270º is from the left. By making these angles relative to the wall, the same profile might be used on different walls around the building without modification depending on building geometry.

To update entered data points on the graph press the "Redraw" button in the curve fit frame. Pressing this button will re-plot the data on the graph and will sort the data points by angle.

Select Curve Fit:

Curve Fit 1: This curve fit simply connects the data points and linearly interpolates between the data points.

Curve Fit 2: This curve fit connects all the user data points using a nonlinear curve fit between the points (cubic spline).

Curve Fit 3: This curve fit requires five data points – one for each wind direction angle 0º, 90º, 180º, 270º and 360º. The curve fits a trend line to these five data points (ignoring any others) according to the following equation developed by Walker and Wilson [Walker and Wilson 1994 and ASHRAE 2005 p 27.6].

$$Cp(\theta) = \frac{1}{2}\left[\left(Cp(0^\circ) + Cp(180^\circ)\right)\left(\cos^2\theta\right)^{\frac{1}{4}} + \left(Cp(0^\circ) - Cp(180^\circ)\right)\left(\cos\theta\right)^{\frac{3}{4}} + \left(Cp(90^\circ) + Cp(270^\circ)\right)\left(\sin^2\theta\right)^2 + \left(Cp(90^\circ) - Cp(270^\circ)\right)\left(\sin\theta\right)\right]$$

Airflow Element Types

This is a list of available airflow elements or mathematical models that provide the relationship between airflow and pressure difference for airflow paths. See Airflow Analysis in the Theoretical Background section of the manual for mathematical details.

One-way Flow using Powerlaw Models

These one way flow models permit flow in the direction of the pressure drop. CONTAM provides you with the following powerlaw flow models.

1. Q = C(ΔP)^n: This is the general form of the powerlaw model in volumetric flow form that allows you to directly input the coefficient, *C*, and exponent, *n*.
2. F = C(ΔP)^n: This is the general form of the powerlaw model in mass flow form that allows you to directly input the coefficient, *C*, and exponent, *n*.
3. Leakage Area Data: This model refers to effective leakage areas as described in Chapter 26 of the *2001 ASHRAE Handbook of Fundamentals*.
4. Connection (ASCOS) Data: Refers to the airflow description used in the ASCOS program [Klote 1982].
5. Orifice Area Data: Relates the opening description to the orifice area data.
6. Crack Description: A narrow opening described by its length and width.
7. Test Data (1-point): Uses a single flow rate and pressure drop along with an estimate for the pressure exponent, *n*.
8. Test Data (2-points): Uses two flow rates with their corresponding pressure drops to define the flow using the powerlaw model.
9. Stairwell: Data corresponding to a stairwell is fit to the powerlaw model.
10. Shaft: Describes the flow in a large shaft.

One-way Flow using Quadratic Models

These models permit flow in only one direction through the path based on the pressure drop.

1. ΔP = aQ + bQ^2 and ΔP = aF + bF^2: These models allow you to directly input the "a" and "b" coefficients for the Quadratic models.
2. Crack Description: This model describes a narrow opening in greater detail than the powerlaw version of the crack model.

3. Test Data (2- point): This model uses two flow rates and their corresponding pressure drops to define the flow using the quadratic model.

Two-way Flow Models

These models are used to represent tall openings where temperature differences across the opening may cause two-way flow across the opening, e.g., doorways.

1. Single Opening Model with 2-way Flow: This models the flow through a path as a single large opening.
2. Two-opening Model: This model divides an opening vertically and models the flow using two powerlaw models, one for each opening. Results are obtained in the form of the net flow rate in each direction.

Backdraft Damper Flow Models

The models allow you to model a feature that has different resistances depending on the direction of the pressure drop, e.g., a smoke control damper. CONTAM implements both a volume flow and a mass flow version of the backdraft damper model, both of which are based on the powerlaw model.

1. Backdraft Damper Model: F = C(ΔP)^n: This is the volumetric flow form of the Backdraft Damper model. You input separate values of C and n depending on the sign of the pressure drop across the damper.
2. Backdraft Damper Model: Q = C(dP)^n: This is the mass flow form of the Backdraft Damper model. You input separate values of C and n depending on the sign of the pressure drop across the damper.
3. Self-regulating Vent: This element limits the airflow rate in both directions through a flow path with user-defined limiting air pressure differences. CONTAM implements the self-regulating vent as described in [Axley 2001].

Fan and Forced Flow Models

These models allow you to create airflow paths that represent fans or other paths for which you want to designate a flow rate.

1. Constant Mass Flow Fan: This model provides for a constant mass flow rate.
2. Constant Volume Flow Fan: This model provides for a constant volumetric flow rate, but the mass flow will vary based on the actual density.
3. Cubic Polynomial Fan Model: This model implements a user-defined fan performance curve to relate pressure vs. flow rate.

Cubic Spline Flow Models

These models provide the ability to create user-defined airflow elements based on a set of data points to which a cubic spline curve is fit. There are four versions that relate either mass flow or volume flow to pressure difference across the flow element (or vice versa). The cubic spline curve fit insures that the relationship between flow and pressure is differentiable as required by the CONTAM airflow solver.

1. F vs P: *Mass flow rate* as a function of *Pressure difference*
2. Q vs P: *Volume flow rate* as a function of *Pressure difference*

3. P vs F: *Pressure difference* as a function of *Mass flow rate*
4. P vs Q: *Pressure difference* as a function of *Volume flow rate*

One-way Flow Using Powerlaw Models

CONTAM performs airflow calculations using the mass flow powerlaw formula, F = C(ΔP)^n, for the following types of airflow elements. Where F is the mass flow rate, ΔP is the pressure difference across a flow path, C is the flow coefficient and n is the flow exponent. However, for some of the powerlaw models, ContamW enables you to input parameters that describe an opening in terms other than the coefficient C and exponent n. These parameters are then converted internally by ContamW to determine C and n for the mass flow version of the powerlaw model.

Powerlaw Model: Q = C(ΔP)^n

This model allows you to directly enter the coefficient C and exponent n for the volume flow version of the powerlaw model.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Flow Coefficient (C): The coefficients may only be expressed in SI units due to the conversion method used. Use the following conversion to convert from IP units to SI units.

To convert from units of cfm/in H_2O^n to the SI units of m3/s Pan

$$\text{To convert from units of } \frac{cfm}{in\,H_2O^n} \text{ to SI units of } \frac{m^3/s}{Pa^n} \text{ multiply by } \frac{1}{2119 \times 249^n}$$

Flow Exponent (n): Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Powerlaw Model: F = C(ΔP)^n

This model allows you to directly enter the coefficient C and exponent n for the mass flow version of the powerlaw model.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Flow Coefficient (C): The coefficients may only be expressed in SI units due to the conversion method used. Use the following conversion to convert from IP units to SI units.

$$\text{To convert from units of } \frac{lbm/s}{in\,H_2O^n} \text{ to SI units of } \frac{kg/s}{Pa^n} \text{ multiply by } \frac{0.4536}{249^n}$$

Flow Exponent (n): Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Powerlaw Model: Leakage Area

Leakage area refers to a description of airflow features given in Chapter 26 of the 2001 ASHRAE Handbook of Fundamentals [ASHRAE 2001]. Table 1 of this reference provides typical leakage areas for residential buildings.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Leakage Area: There are three possible ways to enter the leakage area: per item, per unit length, and per area. Select the radio button for the type of leakage area to be described, then enter the appropriate value for the type of leakage selected.

Per Item: Enter a total leakage value for an item, this is usually used for a doorway or window - something that can be classified as an item.

Per Unit Length: Commonly used to describe an interface such as a wall/ceiling junction.

Per Unit Area: Used to describe an area such as a wall or floor.

Reference Conditions: Be sure to check the reference condition for the reported leakage areas. Two sets of reference conditions are common [ASHRAE 2005 p 27.13]:
Discharge coefficient of 1.00 at a reference pressure difference of 4.0 Pa and
Discharge coefficient of 0.611 at a reference pressure difference of 10 Pa.

Discharge Coefficient: Enter the discharge coefficient for the leakage area at the reference pressure difference.

Flow Exponent: Enter the flow coefficient for the leakage area at the reference pressure difference. The flow exponent is not reported and therefore must be estimated. For openings associated with infiltration, measurements usually indicate an exponent between 0.6 and 0.7.

Pressure Difference: Enter the reference pressure difference for the associated leakage rating.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Powerlaw Model: Connection (ASCOS)

The ASCOS connection element is provided for compatibility with the ASCOS program [Klote 1982]. It is an implementation of the more general orifice flow element based upon the orifice equation.

$Q = K A_o (2 \Delta P / \rho)$

The inputs for the ASCOS connection are the opening area, A_o, and the dimensionless flow coefficient, K. Data describing a connection are reduced to the powerlaw model with an exponent of 0.5. The orifice flow element (see Powerlaw Model: Orifice Area) provides a more general implementation.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Flow Coefficient: The flow coefficient *K* 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: Ao, refers to the observable area of the opening.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Powerlaw Model: Orifice Area

This airflow element allows you to input the description of an orifice.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Cross-sectional Area: This refers to the observable area of the opening.

Flow Exponent (n): Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Discharge Coefficient: The discharge coefficient, C, is related to the dynamic effects and is typically close to 0.6 for a sharp-edged orifice and slightly higher for other openings in buildings.

Hydraulic Diameter: The hydraulic diameter is equal to (4 · Area / Perimeter). For square openings this equals the square root of the area, and for long thin openings it is two times the width.

Reynolds Number: The transition from laminar flow to turbulent flow occurs over a very broad range of Reynolds numbers with the flow being fully laminar approximately below 100.

Note: The hydraulic diameter and Reynolds number have little impact on the calculations. Generally you should use the default values except for special circumstances where you need them to be modified. The parameters above describe the flow characteristics of an orifice in typical operation. At extremely low pressure drops the use of the powerlaw model leads to a division by zero during the network solution process. ContamX avoids this problem by changing to a linear model in this region. The model is based conceptually on the flow changing from turbulent to laminar at very low pressures. The Hydraulic diameter and Reynolds number are used to determine a point where the model changes from the powerlaw to linear.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Powerlaw Model: Crack Description

This airflow element allows you to enter the description of a crack and is based upon the relationships developed between flow through cracks and the powerlaw model [Clark 1985].

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Crack Length/Width: Enter the length and width of the crack.

Description: Field for entering a more detailed description of the specific airflow element.

Note: ContamW automatically uses the icon for a small opening for the powerlaw model crack description.

Powerlaw Model: Test Data (1 point)

This airflow element allows you to provide a single test data point that ContamW reduces to the powerlaw model by calculating the flow coefficient based on input values (See Fitting Powerlaw Coefficients in the Theoretical Background section).

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Pressure Difference (ΔP): Enter the pressure difference for the data point.

Flow Rate (F): Enter the flow rate for the data point.

Pressure Exponent (n): The flow exponent is a guess used to fit the single point to the powerlaw model. A value of 0.6 to 0.7 is typical for an infiltration element. A value of 0.5 is more appropriate for large openings.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Powerlaw Model: Test Data (2 points)

This airflow element allows you to provide two test data points that ContamW reduces to the powerlaw model by calculating the flow coefficient based on input values (See Fitting Powerlaw Coefficients in the Theoretical Background section).

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Pressure Difference (ΔP): Enter the pressure difference for each of the two data points.

Flow Rate (F): Enter the flow rate for each data point.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Powerlaw Model: Stairwell

This airflow element allows you to enter a description of a stairway, which ContamW converts to a powerlaw relationship assuming a pressure exponent of 0.5. A stairwell will normally be

modeled as a vertical series of zones connected by low resistance openings (this stairwell flow element) through the floors. The resistance is based on a fit to experimental data [Achakji & Tamura 1998]. This model should only be used between zones on different levels of a building (paths through floors), not between zones on the same level (paths through walls).

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Distance Between Levels: The vertical distance between doorways of the stairwell. This is typically equal to the "Distance to level above" of the level below the path with which this element is to be associated.

Cross-sectional area: This is the horizontal, cross-sectional, area of the shaft.

Density of People: A large number of people in the stairwell influences the flow resistance of the stairwell. The experiment used densities of 1, 2, and 3 people per square meter. These are the only units available in ContamW. The following conversion is provided for your convenience.

1 m^2 = 10.76 ft^2

Stair Treads: There are two options for this field: open tread and closed tread. This refers to the front of the tread whether or not the tread is open effects flow.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Powerlaw Model: Shaft

This airflow element allows you to enter a description of a shaft, and ContamW converts the information to a powerlaw relationship assuming a pressure exponent of 0.5. A shaft will normally be modeled as a vertical series of zones connected by low resistance openings (this shaft airflow element) 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 [ASHRAE 2005, p2.7].

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Distance Between Levels: This is the distance between openings for the stairwell.

Cross-sectional area: This is the horizontal, cross-sectional, area of the shaft, not the opening.

Perimeter: The perimeter of the horizontal cross-section of the shaft. This number is used in conjunction with the area to create the hydraulic diameter.

Roughness: This refers to the average size of the protrusions from the shaft wall into the airflow.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

One-way Flow Using Quadratic Flow Models

ContamX performs airflow calculations using the mass flow quadratic relationship ($\Delta P = aF + bF^2$) between mass flow rate and pressure difference across a flow path for the following types of airflow elements. Where F is the mass flow rate, ΔP is the pressure difference across a flow path, and a and b are flow coefficients.

Quadratic Model: $\Delta P = aQ + bQ^2$

This model allows you to directly enter the coefficients, a and b, into the volume flow version of the quadratic flow model. Where ΔP is the pressure drop and Q is the volumetric flow rate.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Coefficients: The coefficients must be expressed in SI units because of the way ContamW handles unit conversions. The units of the coefficients at standard conditions, i.e., sm^3/s, are as follows:

a [$Pa \cdot s/sm^3$]

b [$Pa \cdot (s/sm^3)^2$]

To convert from volume flow Q in units of m^3/s at density ρ to sm^3/s at density ρ_s, multiply Q by ρ/ρ_s, where ρ_s equals 1.2041 kg/m3 at 20 °C.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Quadratic Model: $\Delta P = aF + bF^2$

This model allows you to directly enter the coefficients, a and b, into the mass flow version of the quadratic flow model. Where ΔP is the pressure drop and F is the mass flow rate.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Coefficients: The coefficients must be expressed in SI units because of the way ContamW handles unit conversions. The units of the coefficients are as follows:

a $\left[Pa \cdot s/kg \right]$
b $\left[Pa \cdot (s/kg)^2 \right]$

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Quadratic Model: Crack Description

This model employs a quadratic relationship of the form $\Delta P = aQ + bQ^2$ where Q is the volume flow rate [Baker, Sharples, and Ward 1987]. CONTAM uses the mass flow version of that formula: $\Delta P = aF + bF^2$ where ΔP is the pressure drop and F is the mass flow rate.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Crack Dimensions:

Length: The overall length of the crack

Width: Width of the crack

Depth: The distance along the direction of airflow

Number of Bends: The number of bends in the flow path.

Description: Field for entering a more detailed description of the specific airflow element.

Quadratic Model: Test Data (2 points)

Measured data (2 points) is reduced to the mass flow version of the quadratic model.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Data: Data consists of two sets of pressure drops and corresponding flow rates.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Two-way Flow Models

These models enable you to simulate openings through which two-way airflow might occur (e.g. doorways and open windows). When viewing results of these models, the bidirectional flow will be indicated on the SketchPad if it occurs. You can also use ContamW's airflow plotting feature to plot both airflows or to simply plot the net airflow. There must be a density difference across the associated airflow path in order for bidirectional flow to occur across these types of airflow elements. See Doorways (Large Openings) in the Theory section for a more detailed explanation.

The *Relative Elevation* of flow paths that implement these models should be set to the bottom of the flow path as opposed to the mid-height used for other element types.

Two-way Flow Model: Single Opening

This is a model for flow through large openings such as doorways through which air can flow in two directions simultaneously throughout the opening.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Height: The overall height of the opening, not to be confused with the *Relative Elevation* of flow paths with which the element is associated.

Width: The width of the opening. Again, before entering the dimension make sure to select the proper units from the drop down menu.

Flow Coefficient: An experimentally determined value. Experiments by Weber and Kearney [Weber and Kearney 1989] have shown the default value of 0.78 to work well for most applications.

Minimum Temperature Difference for Two-Way Flow: A two-way flow is driven by the temperature (actually air density) difference between the two zones. When this temperature difference approaches zero the algorithm used for solving the flow tends towards a division by zero problem. To avoid this undefined situation the two-way model reverts to a one-way power law model at this "minimum temperature difference" using the opening size to define the orifice at ΔT set in this field. ContamW uses a default value of 0.01 °C.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Two-way Flow Model: Two-opening

This model accounts for two-way flow due to the stack effect acting over the height of a tall opening. It uses two power law flow models at different heights to approximate a single tall opening.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Height: The overall height of the opening, not to be confused with the *Relative Elevation* of flow paths with which the element is associated.

Width: The width of the opening.

Flow Coefficient: This is an experimentally determined value. Experiments by Weber and Kearney have shown the default value of 0.78 to work well for most applications.

Description: Field for entering a more detailed description of the specific airflow element.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Backdraft Damper Models

The Backdraft Damper models make it possible to model a feature that has different flow resistances depending on the direction of the pressure drop, with greatly reduced (or zero) flow in one direction. Note that there are similar models available for duct flow elements.

Backdraft Damper Model: Q = C(ΔP)^n

This is the volumetric flow form of the backdraft damper airflow model.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Pressure Difference: Enter a flow coefficient and exponent for both the positive and negative flow directions of the flow path. **ΔP>0** refers to the pressure difference across the flow element that would lead to an airflow in the positive flow direction (See Airflow Path Properties) and **ΔP<0** would lead to a flow in the opposite direction.

Flow Coefficients: The coefficients may only be expressed in SI units due to the conversion method used. Use the following conversion to convert from IP units to SI units.

To convert from units of $\frac{cfm}{in\,H_2O^n}$ to SI units of $\frac{m^3/s}{Pa^n}$ multiply by $\frac{1}{2119 \times 249^n}$

Flow Exponents: Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Description: Field for entering a more detailed description of the specific airflow element.

Backdraft Damper Model: F = C(ΔP)^n

This is the mass flow form of the backdraft damper airflow model.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Pressure Difference ΔP>0 or ΔP<0: This indicates the direction of predominant flow for the model.

Flow Coefficients: The coefficients may only be expressed in SI units due to the conversion method used. Use the following conversion to convert from the IP units to SI units.

To convert from units of $\frac{lbm/s}{in\,H_2O^n}$ to SI units of $\frac{kg/s}{Pa^n}$ multiply by $\frac{0.4536}{249^n}$

Flow Exponents: Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Description: Field for entering a more detailed description of the specific airflow element.

Self-regulating Vent Model

CONTAM implements the self-regulating vent as described in [Axley 2001]. This element limits the airflow rate in both directions through a flow path with user-defined limiting air pressure differences.

Flow is calculated based on the following equations for positive and negative pressure differences across the element. The positive flow direction is set for each airflow path.

$$Q^+ = Q_0\left(1.0 - e^{\frac{-\Delta P}{\Delta P_0}}\right)$$

$$Q^- = -fQ_0\left(1.0 - e^{\frac{\Delta P}{f\Delta P_0}}\right)$$

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Maximum Flow Rate (Q_0) - an empirical value that sets the maximum airflow rate that will be allowed to pass through the airflow path.

Regulating Pressure (ΔP_0) - an empirical value that represents the approximate pressure difference above which the airflow will be limited to the maximum flow rate Q_0.

Reverse Flow Fraction (*f*) - the fraction of the Maximum Flow Rate to which the airflow through this element will be limited when the pressure difference is negative across the airflow path.

Description - You can provide a more detailed note to describe this airflow element.

Fan and Forced Flow Models

These airflow element models enable you to easily create forced airflow elements as airflow paths between two zones, as opposed to implementing a duct model or simple air handling system. There are three types of fan models to choose from: constant mass flow rate, constant volumetric flow rate, and fan performance curve.

Fan Model: Constant Mass Flow Fan

This model describes an airflow element having a constant mass flow rate. This airflow element will provide the specified constant mass flow rate regardless of the density of the air being delivered by the fan.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Design (maximum) Flow: Enter the maximum mass flow rate. This value can be modified by the path schedule.

Description: Field for entering a more detailed description of the specific airflow element.

Fan Model: Constant Volume Flow Fan

This model describes an airflow element having a constant volume flow rate. The design flow rate you specify is in terms of standard air. CONTAM uses a density of 1.204 kg/m^3 based on conditions specified by ASHRAE for dry air: 101.325 kPa and 20 ºC (ASHRAE 2004, p 18.4). Therefore, if actual conditions during simulation do not match these standard conditions, the results will differ from specified flow due to differences in air density.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Design (maximum) Flow: Enter the maximum volume flow rate. This value can be modified by the path schedule.

Description: Field for entering a more detailed description of the specific airflow element.

Fan Model: Fan Performance Curve

This airflow element allows you to create a fan based on a fan performance curve. You provide ContamW with a set of pressure rise and airflow rate data points, and ContamW performs a polynomial curve fit to the data. Airflow rates are specified in mass flow units for standard air.

Name: Enter the name you want to use to identify the airflow element. The airflow element will be saved within the current project and can be associated with multiple airflow paths.

Fan off conditions: These values specify how the fan is modeled when the speed of the fan is insufficient to provide a flow.

> **Cut-off Ratio:** The *fan speed ratio* is the actual speed (RPM) of the fan divided by the rated speed of the fan (fan is defined for a speed ratio of 1.0). When this ratio is below the

specified cut-off limit the fan is modeled as a simple orifice based on the "equivalent orifice" defined for this fan element. During a simulation, ContamX obtains the actual ratio from the control value being applied (in the form of a schedule or control signal) to the airflow path with which this fan element is associated and compares it to the *cut-off ratio*. See Chapter 18 Fans in [ASHRAE 2004].

Equivalent Orifice: The size of the orifice used to model the fan opening when the control value drops below the specified limit.

Description: Field for entering a more detailed description of the specific airflow element.

Fan Performance Data: Click the "Edit Fan Curve Data" to input/modify the fan curve for this airflow element. A detailed explanation of this data is given in the next section, Fan Performance Data.

Fan Performance Data

Between four and ten data points are required for ContamW to fit a cubic polynomial to create a performance curve for the fan. The fitted curve may not contain a point of contraflecture; if it does an error message will be displayed. If this error message is displayed the pressure rise data must then be modified to eliminate the point of contraflecture.

Entering Fan Curve Data Points

You only need to fill in the "Flow rate" and "Pressure rise" data entry when entering fan curve data. When adding a new data point, the flow rate must be entered in the "Flow rate" edit box and the pressure rise must be entered in the "Pressure rise" edit box. Use the "Revised dP" box when making changes to a pressure rise that has already been entered. This will preserve the original data point but use the revised value when performing the curve fit. After entering the flow rate and the pressure rise into the appropriate edit boxes, the next step is to press the "<< Insert <<" button. Once you have entered four data points a cubic fit will automatically be generated and displayed as a line on the graph.

Revising Fan Curve Data Points

To edit existing data points, highlight the line of data to revise in the list of Fan Curve Data to the left of the Insert, Replace and Delete buttons. The values from that data set will then appear in the edit boxes to the right of the buttons.

If you need to modify the flow rate, enter the new flow rate in the "Flow rate" edit box and then press the "<< Replace <<" button. The new data will now replace the old data.

If the Pressure Rise needs to be modified there are two available options:

1. Replace the data point completely. To overwrite an existing pressure rise and refit the fan curve, select the data set to be modified by highlighting it in the data list and then type the new pressure rise in the "Pressure rise" edit box then press the "<< Replace <<" button.
2. Change the point used to fit the cubic polynomial but leave the marker for the original data point intact. With this option you may "tweak" the curve in the event there is some issue with the originally entered fan curve, such as a point of contraflecture.

Cubic Spline Models

Cubic spline models enable you to create airflow elements based on a curve fit to a user-defined set of data points. The cubic spline fit used to generate the curve guarantees a first-order differentiable relationship between flow and pressure as required by the CONTAM solver. During simulation, the sign of the pressure difference will be based on the *Positive Flow Direction* defined for the flow paths with which the spline elements are associated. For example, a drop in pressure in the direction of positive flow (i.e. Pressure in from zone – Pressure in to zone > 0) will utilize a positive pressure from the spline data.

There are some basic requirements for each of these elements. They require a minimum of four data points, so a curve will not be displayed until the minimum number of points is entered. All models require that the slope be greater than zero for all segments of the curve fit. If there is an error in the curve fit when you click the OK button, an error message will be displayed indicating the offending segments of the curve and a reason for the error. Segments are numbered from zero to the number of data points minus one. For example – "`seg 0: y′ <= 0`" will be displayed if the segment between the first two data points has a slope less than or equal to zero.

Name: Enter the name you want to use to identify the airflow element.

Description: Field for entering a more detailed description of the specific airflow element.

Curve Data: Create and edit the list of data points to define the curve for the airflow element. The type of cubic spline element you are editing determines the independent and dependent variables for the curve. The labels of the list, data entry fields and associated units will change accordingly, as will the axes of the plot. Use the data entry fields along with the "Add" button to create new and edit existing data points. Use the "Delete" button to remove the currently selected data point from the list.

Icon: Choose either the small or large opening icon as appropriate for the specific airflow element. The icon has no effect on the simulations.

Cubic Spline: F vs P

Mass flow as a function of Pressure drop across the element.

Cubic Spline: Q vs P

Volume flow as a function of Pressure drop across the element.

Cubic Spline: P vs F

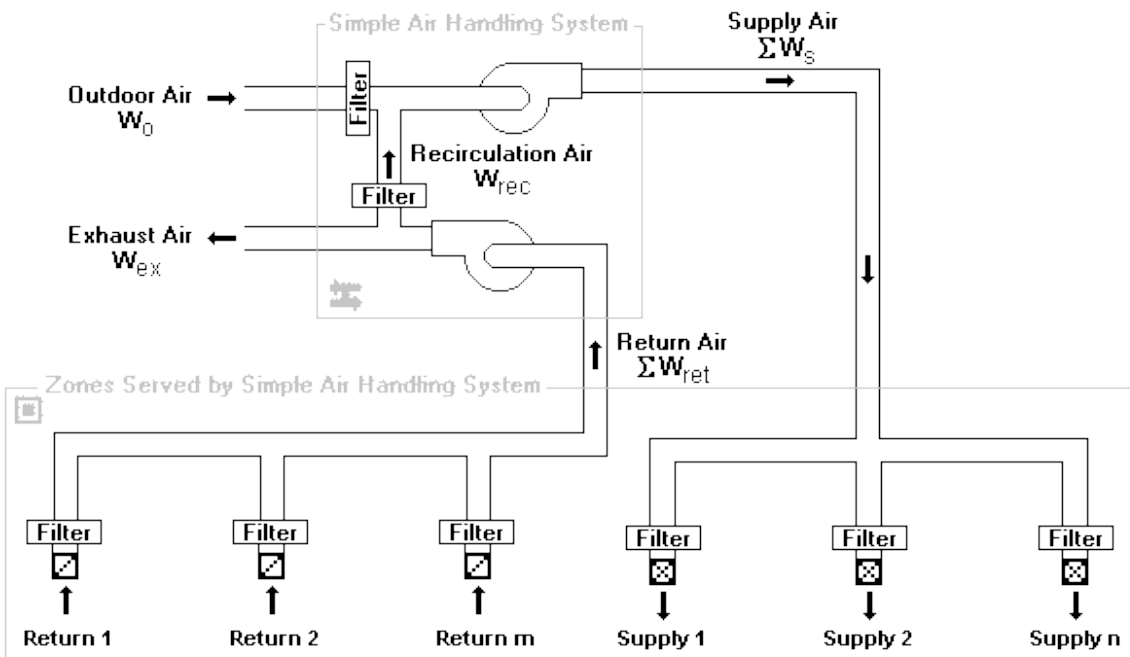
Pressure drop as a function of Mass flow through the element.

Cubic Spline: P vs Q

Pressure drop as a function of Volume flow through the element.

3.6 WORKING WITH SIMPLE AIR-HANDLING SYSTEMS

The *simple air-handling system* (AHS) provides a convenient means of incorporating an air-handling system into a building without having to draw and define an entire duct system. Each simple air-handling system consists of two implicit airflow nodes or zones (*supply* and *return* sub-systems), three implicit flow paths (*recirculation*, *outdoor*, and *exhaust*), and multiple *zone supplies* (inlets to zones) and *zone returns* (outlets from zones) that you can place within zones throughout the building. You specify the airflow rates of each *supply* and *return* point. *Simple air-handling systems* do not require you to associate both zone supplies and zone returns with them. You may use the *simple air-handling system* to only supply outdoor (ambient) air to a building or to only exhaust air from the building. The following figure shows a schematic representation of a simple air-handling system.



Determination of System Airflows

ContamX determines the airflows rates associated with each air-handling system according to the following algorithm. More detailed explanations for the values that you must input for this algorithm are explained in the Defining Air-handling Systems section that follows.

First, all of the user-defined supply (supply air to the zones) and return (return air from the zones) airflow (mass) rates are summed.

ΣW_s ≡ the sum of all supplies

ΣW_{ret} ≡ the sum of all returns

The amount of outdoor air that the system requires is then determined by

$W'_o = \max(f_o \cdot \Sigma W_s, \min(\Sigma W_s, W_{o_min}))$,

where, f_o ≡ the fraction outdoor air = W_o/W_s

You input this parameter as the *Outside Air Schedule* of a simple air handling system (See Air Handling System – AHS Properties), and

W_{o_min} ≡ the "minimum outdoor airflow" parameter that you input.

The rate at which air is recirculated via the implicit recirculation flow path, W_{rec}, is determined by

$W_{rec} = \min(\Sigma W_{ret}, \Sigma W_s - W'_o)$.

The rate at which outdoor air is brought in by the system via the implicit outdoor airflow path, W_o, is determined by

$W_o = \Sigma W_s - W_{rec}$.

And the rate at which exhaust airflows to the ambient via the implicit exhaust flow path, W_{ex}, is determined by

$W_{ex} = \Sigma W_{ret} - W_{rec}$.

Based on this algorithm the amount of outdoor air the system will provide, W_o, will be between W_{o_min} and $f_o \cdot \Sigma W_s$ as long as the demand for supply air, ΣW_s, is sufficient to provide this value. Otherwise the system will provide ΣW_s. When the sum of the supply flows, ΣW_s, exceeds the sum of the return flows, ΣW_{ret}, the balance is made up of *outdoor* air. Any excess return air is exhausted via the implicit exhaust flow path.

3.6.1 Creating Simple Air-handling Systems

You create *simple air-handling systems* by placing an *Air-handling System (AHS) icon* on the SketchPad along with some combination of *supply* and/or *return icons* (See Drawing Building Component Icons). You may place an *AHS icon* on any blank cell of the SketchPad on any level. You do not have to place it within a zone. However, you must place *Supply icons* and *Return icons* within a zone. The pop-up menu selections will be disabled (grayed out) if you pop-up on a location where ContamW does not allow these icons to be placed.

3.6.2 Deleting Air-handling Systems

You delete icons related to simple air-handling systems using the icon deletion procedure (See Deleting Building Component Icons). If you delete an AHS icon, , all of the supplies and returns associated with that system will be deleted. You will be warned of this when you attempt to delete the icon. However, you can delete individual supplies and returns without affecting any other icons. Deleting any AHS-related icons from the SketchPad does not delete any filter, schedule, or kinetic reaction elements that you may have defined for the AHS. These flow elements can only be deleted using the CONTAM Library Manager, accessible through the **Data** menu for each type of data element.

3.6.3 Modifying Air-handling Systems

To modify the parameters of a simple air-handling system, you use the icon definition procedure (See Defining Building Component Icons) to display its properties and make the desired changes. You can move air-handling system icons and move and copy supply and return icons; however, you cannot copy air-handling system icons.

Moving Air-handling System Icons

You can move AHS icons using the **Cut** and **Paste** functions of the **Edit** menu. ContamW will only allow you to move an AHS icon within the same level upon which it is located.

Moving Supply and Return Icons

Once you define a supply or return icon, you can move them using the **Cut** and **Paste** functions of the **Edit** menu. ContamW will only allow you to move a supply or return icon within the same zone in which it is currently located.

Copying Supply and Return Icons

Once you define a supply or return icon using the **Copy** and **Paste** functions of the **Edit** menu. You can copy supply and return icons into any zone on any level.

3.6.4 Defining Air-handling Systems

After you place an *AHS-related* icon (*AHS*, *Supply* and *Return*) onto the SketchPad you must define them using the icon definition procedure (see Defining Building Component Icons) to display and edit the properties of the icon. This will display the property sheet associated with the icon. Detailed descriptions of all AHS properties are given in the Air-handling System Properties section of this manual.

This list shows the various AHS-related icons and provides a brief description of each.

Icon	Description
	Air-handling system
	Room air supply (inlet) of an air-handling system
	Room air return (outlet) of an air-handling system

Air-handling System Icon

The properties associated with each AHS include outdoor air intake data, supply and return contaminant information, and system filters. Outdoor air intake data consists of a minimum outdoor air intake rate, W_{oa_min} and a schedule for varying the outdoor air intake rate as a percentage of the supply airflow rate. The schedule will set the value of f_o in the algorithm above. The default value of f_o is 1.0 (i.e., 100 % outdoor air) if you don't use a schedule. Supply and return system data refers to the implicit zones that are created for each AHS. Just as for any zone you create, you can provide a zone volume, initial zone concentration and a kinetic reaction for each contaminant defined for the project. You can also associate contaminant filters with the implicit outdoor and recirculation flow paths.

Supply and Return Icons

The properties associated with each AHS include airflow rate and contaminant filtration information. You must enter a design airflow rate for each supply and return icon that you create. Enter the design maximum value that you anticipate requiring. You can also provide a schedule to the design airflow rate to adjust down the design flow rate. As with other airflow paths, you can provide a contaminant filter to these items. Be careful not to place filters in both the individual returns and the recirculation path of the air handling system unless this is what you really intend to do.

3.6.5 Air-handling System Properties

This section provides detailed descriptions of the specific properties of simple air-handling systems. Each of the following subsections are the context-sensitive help topics that you can access by pressing **F1** when working with property pages of the "Simple Air-handling System Properties," "Supply Point Properties," and "Return Point Properties" property sheets.

Air-handling System - AHS Properties

System Name: The name used to identify the Air-handling System. Air-handling system names must be unique.

Minimum OA Flow: This is the minimum *outdoor air* introduced into the AHS *supply* flow. If the sum of the *return* flows plus the *minimum outside airflow* exceeds the total supply flow then the excess return air is exhausted. Similarly if the return flow plus the minimum outdoor airflow do not sum to the supply flow then outdoor air is brought in to make up the difference.

Outdoor Air Schedule: This is a schedule of the fraction of outdoor air, f_o, to be introduced to the simple air handling system. If the schedule is set to "<none>", then the fraction of outdoor air defaults to 100 % (100 %outdoor air, 100 % exhaust and no recirculation). You define a new Outdoor Air schedule by pressing the "New Schedule" button and following the procedure for editing a Week Schedule.

> **Name:** This drop down box contains the names of all previously defined schedules within the current project. You can select an existing schedule from the list and use the "Edit Schedule" button to view and modify its properties, create a "New Schedule" and even select a schedule from a CONTAM Library via the "Library..." button. You can also choose to not apply a schedule by selecting "<none>" from the list.
>
> **Description:** Displays the detailed description of the selected outdoor air schedule.

CVF/DVF Control: Values from either a CVF or DVF file can be used in place of the above Schedule (read about CVF and DVF controls in the Control Element Types section). This enables using these control element types without having to draw/define an associated control network. The associated radio buttons "Use Neither", "Use CVF" and "Use DVF" are used to select the desired control method. Select the heading name from the associated "Name" drop down box that will display the headings of either the CVF or DVF file as per the aforementioned radio button selection.

Control Status: This will display the current method of control: Schedule, direct CVF, direct DVF or Control Network. If an actuator of a control network on the SketchPad is associated with the scheduled item then it will take precedence over the Schedule and CVF/DVF Control. If either the "Use CVF" or "Use DVF" radio button is selected then CVF or DVF will take precedence over an associated schedule.

Outdoor Air Intake Location: These are the absolute X, Y and Z coordinates and units that define the location of the outdoor air intake of the selected air handling system. These values are required when using a *Wind Pressure and Contaminant (WPC)* file (see Working with WPC Files).

NOTE: You must first check either the *Wind Pressures* or *Contaminant Concentrations* check boxes on the *Wind Pressure and Contaminants (WPC) File Parameters* dialog box to be able to access these coordinate input fields (see Create WPC File Dialog).

Air-handling System - Supply System Properties

System Volume: Enter a value for the supply sub-system (implicit supply zone) of the simple air handler (e.g., supply-side duct work etc.). This volume is very similar to a zone volume and will be used in the simulation of contaminant transport.

Contaminant Data: All of the defined contaminants in the project will be displayed in this list box.

Initial Concentration: Select a contaminant from the list and enter the *initial contaminant concentrations* for the *supply air sub-system* of this *simple air handling system*. Note that only those species you have selected to be contaminants (to use during simulation) appear in the list. If a species for which you wish to set an initial concentration does not appear in the list you must set the "Use in simulation" property of the corresponding species to be true (See Creating Species and Contaminants). You can reset these initial values through the Run Control Properties of the Simulation Parameters.

NOTE: Changing the number of contaminants, i.e. those species used in the simulation, will reset the *initial concentrations* of all zones to the default contaminant concentrations. You can reset all of these concentrations via the Run Control Properties as indicated above or individually here.

Sum of Non-trace Initial Concentrations: This value is provided as a check for you to insure that you input a reasonable set of initial values when using *non-trace contaminants*. This value should be very close to 1.0.

Kinetic Reaction: You define a new kinetic reaction by pressing the "New Reaction" button and then completing the kinetic reaction matrix. You can edit an existing matrix by selecting the proper matrix from the drop down box and pressing the "Edit Reaction" button. You must have previously defined contaminants in order to implement a kinetic reaction.

> **Description:** Field that shows a more detailed description of the specific reaction if entered by the user. This description can be modified by pressing the "Edit Reaction" button next to the description field and then changing the description field for the reaction.
>
> **Name:** This contains a list of names of all previously created reactions. The reaction selected from this box will define the behavior of the specific supply system being modified.

Air-handling System - Return System Properties

System Volume: Enter a value for the return sub-system (implicit return zone) of the simple air handler (e.g., return-side duct work etc.). This volume is very similar to a zone volume and will be used in the simulation of contaminant transport.

Contaminant Data: All of the defined contaminants in the project will be displayed in this list box.

Initial Concentration: Select a contaminant from the list and enter the *initial contaminant concentrations* for the *return air sub-system* of this *simple air handling system*. Note that only those species you have selected to be contaminants (to use during simulation) appear in the list.

If a species for which you wish to set an initial concentration does not appear in the list you must set the "Use in simulation" property of the corresponding species to be true (see Creating Species and Contaminants). You can reset these initial values through the Run Control Properties of the Simulation Parameters.

NOTE: Changing the number of contaminants, i.e. those species used in the simulation, will reset the *initial concentrations* of all zones to the default contaminant concentrations. You can reset all of these concentrations via the Run Control Properties as indicated above or individually here.

Sum of Non-trace Initial Concentrations: This value is provided as a check for you to insure that you input a reasonable set of initial values when using *non-trace contaminants*. This value should be very close to 1.0.

Kinetic Reaction: You define a new kinetic reaction by pressing the "New Reaction" button and then completing the kinetic reaction matrix. You can edit an existing matrix by selecting the proper matrix from the drop down box and pressing the "Edit Reaction" button. You must have previously defined contaminants in order to implement a kinetic reaction.

> **Description:** Field that shows a more detailed description of the specific reaction if entered by the user. This description can be modified by pressing the "Edit Reaction" button next to the description field and then changing the description field for the reaction.
>
> **Name:** This contains a list of names of all previously created reactions. The reaction selected from this box will define the behavior of the specific return system being modified.

Air-handling System - Filter Properties

As the names suggest, *outdoor air filters* affect the air brought in by the simple air-handling system from outside the building, while *recirculation air filters* affect the return air being circulated back through the air handler. You may either create, edit or delete the filters associated with the implicit outdoor air and recirculation flow paths of the simple air-handler. Creating a new filter or modifying an existing one will display the Filter dialog box. For an explanation of contaminant filters see Working with Filters in the Using CONTAM section of the manual.

Supply and Return Point - System Properties

Design Flow Rate: Enter the design maximum airflow rate for the supply or return.

AHS: You must associate each supply and return point with an existing simple air-handling system.

> **Name:** This drop down box contains the names of all previously created Air-handling Systems. The AHS selected from this box will define the behavior of the specific supply/return system being modified. Once an existing AHS is selected it may be edited by pressing the "Edit AHS" button.
>
> **Description:** Displays the description of the currently selected Simple AHS.

Location: These are the X, Y and RelHeight (Z) coordinates and units for the selected supply/return point. X and Y are absolute coordinates and RelHeight is relative to the level on which the supply/return point is located.

These values are required for any supply/return point that is located within a *1D convection/diffusion zone* (see 1D Zone Data). These coordinates will be verified by ContamW against the axis of the 1D zone in which it is located. This verification will be performed automatically when you select either the **Run Simulation** or **Create a ContamX Input File** command from the **Simulation** menu.

Supply and Return Point - Schedule Properties

Schedule: You can associate a schedule with the supply/return point to control the fraction of the design airflow rate that flows to/from the zone in which the supply/return point is located according to the time of day and day of the week.

Description: Field that shows a more detailed description of the specific schedule if entered by the user. You can modify this description by pressing the "Edit Schedule" button next to the description field and then changing the description field for the schedule.

Name: This drop down box contains the names of all previously created schedules. The schedule selected from this box will define the behavior of the specific supply/return system being modified.

CVF/DVF Control: Values from either a CVF or DVF file can be used in place of the above Schedule (read about CVF and DVF controls in the Control Element Types section). This enables using these control element types without having to draw/define an associated control network. The associated radio buttons "Use Neither", "Use CVF" and "Use DVF" are used to select the desired control method. Select the heading name from the associated "Name" drop down box that will display the headings of either the CVF or DVF file as per the aforementioned radio button selection.

Control Status: This will display the current method of control: Schedule, direct CVF, direct DVF or Control Network. If an actuator of a control network on the SketchPad is associated with the scheduled item then it will take precedence over the Schedule and CVF/DVF Control. If either the "Use CVF" or "Use DVF" radio button is selected then CVF or DVF will take precedence over an associated schedule.

Supply and Return Point - Filter Properties

Filter: You may select a previously defined filter from those contained in the associated Name list. Click the "New Filter" button to define a new filter. Click the "Edit Filter" button to view or modify existing filter data. Setting this field to <none> indicates no filter for this airflow path. (See Contaminant Filters under Working with Contaminants)

3.7 WORKING WITH DUCTS

As an alternative to the simple air-handling system, CONTAM allows you to model HVAC systems using detailed duct systems. Duct systems are made up of duct segments, junctions, terminal points, and *forced air components*. A *duct segment* is a section of a duct between any two junctions, terminal points or combination of the two. A *terminal point* is an endpoint of a duct segment, and a *junction* is the connection between two to four duct segments. Only horizontal segments can be displayed on the ContamW SketchPad, but vertical segments can be implemented as well. Vertical segments are connected with special junctions and terminal points that reveal the location of the vertical segments on the SketchPad.

You draw the layout of the duct system, and ContamW will insert junctions and terminal points where appropriate. Once you have drawn a duct system, you must define the drawing by providing detailed segment, junction and terminal point information. You must provide each duct segment with specific information that describes the flow characteristics and physical dimensions of that segment. Much of this information is in the form of *duct flow elements* that you create when defining the duct segments. Definition of duct segments, junctions and terminals can be simplified using the copy and paste commands. The details of drawing and defining a duct system are described in the following sections.

3.7.1 Duct Flow Elements

Each duct segment must refer to a *duct flow element*. *Duct flow elements* describe the mathematical relationship between flow through and pressure drop along the duct, the flow resistance or forced flow characteristics, cross-sectional geometry, and optional leakage per unit length of a duct. ContamW provides you with several different mathematical models or element types to choose from. Each of these duct flow element types is described in detail in the Duct Segment Properties section. While every duct segment must refer to a single *duct flow element*, multiple duct segments can refer to the same *duct flow element*. *Duct flow elements* can also be stored within a CONTAM library file and shared between different CONTAM project files.

3.7.2 Drawing Ducts

You draw ducts using the duct drawing tool previously described in the Working with the SketchPad section. After you complete the "finalize drawing object" stage of duct drawing, the completed duct will appear as an undefined set of red duct icons. The color red indicates that the duct is not yet defined. Once you have defined the duct components, they will be displayed in blue. A directional duct segment icon will be displayed within each duct segment, and each junction and terminal point will be replaced with an icon indicating the type of junction or terminal point you have defined.

Drawing Duct Segments

ContamW provides certain constraints when drawing ducts to insure that a valid duct system will be drawn that conforms to the underlying model of nodal equations upon which CONTAM is based. When drawing a duct, you cannot cross over a line that you are currently drawing. Ducts can cross over walls, but you cannot draw ducts over any other building component icons. After each duct segment is drawn, the ends of the segment will be automatically replaced by either an undefined terminal point or junction icon. You can only begin or end a duct segment drawing in an unoccupied SketchPad cell or on an undefined duct icon. For this reason, you are advised to draw the entire duct system, on a given level, before you define the individual duct segments. If

you cross over a previously drawn duct segment, either defined or undefined, while drawing a new duct segment, no junction will be placed where the two segments cross.

Vertical duct segments do not appear directly on the SketchPad, because the SketchPad displays only plan-view drawings. Vertical duct segments are associated with vertically connected junctions or terminal points. These vertical junction and terminal icons provide access to the properties of the vertical segments that are located below the level upon which the vertical junction or terminal appears.

Drawing Duct Junctions

Undefined horizontal duct junction icons are drawn automatically at the intersections of the ducts after you finalize drawing a duct. Each time you begin or end drawing a duct segment upon an undefined duct segment, junction or terminal point, ContamW will automatically provide an undefined junction icon if there is not yet one at the junction location.

All vertical junctions are connected to vertical duct segments that are located below the level upon which the vertical junction appears. You can create a vertical junction that is isolated from other ducts on a level that is connected either up, down or both up and down to ductwork on adjacent levels. To do this, finalize the drawing object immediately after setting the initial location of the duct object. To do this, you simple press **LMB** (double click) or ↵ twice at the desired location on the SketchPad when you have the duct drawing tool selected. This will place a single undefined terminal point icon within the SketchPad cell. You can later define this undefined terminal point icon as the specific type of junction that you need and associate it with a vertical duct segment.

Drawing Duct Terminal Points

Undefined horizontal terminal point icons are automatically drawn by ContamW at the end of each duct segment that does not end on another previously drawn duct icon.

You can also create terminal points that are isolated from other ducts on a level but connected either up or down to ductwork on an adjacent level by finalizing the drawing object immediately after setting the initial location of the duct object. To do this, you simply press **LMB** (double click) or ↵ twice at the desired location on the SketchPad when you have the duct drawing tool selected. This will place a single undefined terminal point icon within the SketchPad cell. You can later define this terminal point as the specific type that you need and associate it with a vertical duct segment.

Coloring Duct Systems

You can use the *Duct System Coloring Tool* to automatically color sections of the duct system that are associated with fans (forced flow elements) within the duct. You access the *Duct System Coloring Tool* via the **View → Duct Coloring...** menu command. This will display the Duct System Coloring dialog box that contains a list of all duct segments that contain fans (forced flow elements). Select the fan whose ducts you wish to color. Set the inlet side, outlet side and fan segment colors, then click the Set Colors button to have ContamW trace the duct system on the inlet and outlet sides of the fan segment and set the colors.

Use the **View → Color Ducts** menu command to toggle the colors in which ducts are displayed on the SketchPad between the custom duct colors and the default color.

3.7.3 Deleting Ducts

Deleting currently defined ducts is a two-step process. That is, if you want to delete a duct segment, junction or terminal, you must first undefine the item. *To undefine a duct segment* you must highlight the special duct segment icon (that indicates the positive flow direction) and press the **Delete** key or select Delete from the Edit menu. This will highlight the entire duct segment from its two end-points and prompt you to confirm the undefine operation by displaying a message box. Once you undefine a duct segment, it will again be displayed in red and the special duct segment icon will be converted back to a straight segment icon. *To delete the duct segment* you highlight any portion of the undefined segment or associated undefined terminal and perform the deletion. The entire segment to be deleted will be highlighted, and you will be asked to confirm the deletion.

There are several items you should consider when deleting ducts. You cannot delete an undefined junction that is still connected to a defined duct segment. If you delete a junction that is between two undefined segments, the junction will be removed to form a single segment. You cannot delete a terminal without deleting its associated segment.

Deleting the ducts defined on the SketchPad does not delete any duct flow elements that you may have defined. These flow elements can only be deleted using the CONTAM Library Manager, accessible through the **Data** menu for each type of data element.

3.7.4 Modifying Ducts

In order to modify the layout of a duct system, you must undefine any portions to which you want to connect a new duct (i.e. form a junction). To modify the parameters of a duct segment, junction or terminal, you use the icon definition method to display its properties and make the desired changes. You do not have to undefine the item to change its properties, only to change the physical layout.

To remove a junction from between two segments, you should first undefine the two duct segments, remove the junction and then redefine the newly formed longer segment. To split a segment into two segments, double-click the **LMB** with the duct drawing tool on the undefined segment where you wish to place the junction.

3.7.5 Defining Ducts

After you draw a duct, you must define each duct segment, junction, and terminal point. Each of these duct components is defined using the icon definition procedure (See Defining Building Component Icons) to display and edit the properties of the component.

Defining Duct Segments

You define each duct segment by using the icon definition procedure on any portion of an undefined duct segment. This will display the "Duct Segment Properties" property sheet. Detailed descriptions of all duct segment properties are given in the Duct Segment Properties section of this manual. Once you have defined the properties of a duct segment, a special icon will be displayed indicating the positive flow direction of the duct segment (see Directional Duct Segment Icons). From now on, you use this icon to access the properties of the duct segment.

When defining a duct segment, you must associate the segment with a duct flow element. CONTAM combines duct flow element data with segment specific data such as length and

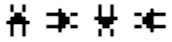
dynamic losses to determine the frictional resistance, volume, and leakiness of a particular duct segment.

You can also define contaminant filtering properties of duct segments. You can define a filter for each contaminant contained in your CONTAM project. A duct filter could be used, for example, to simulate the deposition of particles on the inside surface of a duct.

As previously mentioned, duct flow elements contain duct leakage information. The CONTAM model implements all leakage at the junctions and terminals of a duct segment. This means that half of the leakage associated with a duct segment occurs at each end of the segment. The leakage between a junction and the zone in which the junction is located (as determined on the SketchPad) is a function of the duct element leakage characteristics and the pressure difference between the junction and the zone. You should consider this leakage model when accounting for leakage of a duct that passes through multiple zones. You should put a least one junction (or terminal) in each zone within which you want to account for duct leakage.

Directional Duct Segment Icons

These are the icons that indicate a defined duct segment. The direction that the small arrow points indicates the positive flow direction of the duct segment.



Defining Duct Junctions and Terminals

You define each duct junction and terminal by using the icon definition procedure on a duct junction or terminal icon. This will display the "Duct Junction Properties" property sheet. If a terminal icon is located in the ambient zone, a Wind Pressure property page will be displayed to allow you to account for the effects of wind pressure on the exterior terminal. The Wind Pressure property page will not be displayed for junction and terminal icons located within non-ambient zones. Detailed descriptions of duct junction properties are given in the Duct Junction and Terminal Properties section of this manual. Once you have defined the properties of a duct junction, a special icon will be displayed indicating the specific type of junction you have selected (see the following list). From now on, you use this icon to access the properties of the duct junction.

Junction and Terminal Icons

Icon	Description
	Junction connected to ducts on the same level
	Junction connected to the vertical duct of a junction on the level above
	Junction and downward vertical duct connected to a junction on the level below
	Junction and downward vertical duct connected to a junction on the level below and connected to the vertical duct of a junction on the level above
	Terminal connected to a duct on the same level
	Terminal connected to the vertical duct of a junction on the level above
	Terminal and downward vertical duct connected to a junction on the level below

3.7.6 Duct Segment Properties

This section provides detailed descriptions of the specific duct segment properties. Each of the following subsections are the context-sensitive help topics that you can access by pressing **F1** when working with property pages of the "Duct Segment Properties" property sheet.

Duct Segment - Flow Element Properties

Duct Segment Number: This is the number that appears in the status bar when you highlight a duct segment icon. ContamW automatically assigns this number to each *duct segment* once they are defined. This number could change as you add and remove duct segments to or from a project. Each time you save a project, the duct segment numbers are reassigned. Numbers are assigned beginning on the top level, starting from the upper-left corner of the SketchPad, moving left-to-right and top-to-bottom of the SketchPad.

Duct Flow Element Name: You may select a previously defined airflow element from those contained in the Name list. Click the "New Element" button to define a new airflow element. To view or modify airflow element data click the "Edit Element" button. Every duct segment must have an airflow element associated with it. This field may not be left blank.

Model Summary: This summarizes the information associated with the duct flow element currently displayed in the Name field.

Duct Segment - Segment Properties

Segment Data:

> **Duct segment length:** Enter the length of the duct segment. This will be used along with the cross-sectional area to determine the volume of the duct segment.
>
> **Sum of loss coefficients:** This is the sum of all dynamic loss coefficients due to junction losses and all fittings in the segment. This term only applies to the Darcy-Colebrook duct flow element model (see Ducts in the Airflow Analysis section).
>
> **Positive Flow Direction:** In the case of fans and backdraft dampers it is necessary to know which direction is defined as positive. The arrows represent the positive direction for pressure drop and airflow.

Color: You can set the color of each segment individually here, or you can use the automated *Duct System Coloring Tool* to color sections of ducts related to forced flow elements within the duct system. You access the Duct System Coloring Toolvia the **View → Duct Coloring...** menu command (See Drawing Ducts).

Segment Summary Information: This field displays information about the selected duct flow element.

Duct Segment - Schedule Properties

The schedule value will operate proportionally on various items depending on the type of flow element associated with the duct segment. For most flow elements, it will operate directly on the flow rate. For the fan performance curve element it will act as the ratio of the fan operating speed to the fan off operating speed. The schedule can not be applied to the Darcy-Colebrook element, and the schedule properties will be disabled in the ContamW interface for this type of flow element.

Schedule: If there are previously defined schedules within the current project, you may select one from the Name list. Click the "New Schedule" button to define a new schedule. To view or modify an existing schedule click the "Edit Schedule" button. Setting this field to <none> indicates no schedule. (See Working with Schedules)

CVF/DVF Control: Values from either a CVF or DVF file can be used in place of the above Schedule (read about CVF and DVF controls in the Control Element Types section). This enables using these control element types without having to draw/define an associated control network. The associated radio buttons "Use Neither", "Use CVF" and "Use DVF" are used to select the desired control method. Select the heading name from the associated "Name" drop down box that will display the headings of either the CVF or DVF file as per the aforementioned radio button selection.

Control Status: This will display the current method of control: Schedule, direct CVF, direct DVF or Control Network. If an actuator of a control network on the SketchPad is associated with the scheduled item then it will take precedence over the Schedule and CVF/DVF Control. If either the "Use CVF" or "Use DVF" radio button is selected then CVF or DVF will take precedence over an associated schedule.

Duct Segment - Filter Properties

Filter: You may select a previously defined filter from those contained in the associated Name list. Click the "New Filter" button to define a new filter. Click the "Edit Filter" button to view or modify existing filter data. Setting this field to <none> indicates no filter for this airflow path. (See Contaminant Filters under Working with Contaminants)

Duct Flow Element - Shape, Size and Leakage

Duct Shape: You can implement several different shapes of ducts including: round, rectangular, oval or other. Select the shape you want and the appropriate dimensions will be made available for you to enter under the Duct Dimension parameters.

Duct Dimensions: The data entered here is dependent on the duct shape. For circular ducts, the diameter is the only parameter needed. Rectangular ducts require you to enter the width and height, and oval ducts require you to input the major and minor dimensions. The other option allows you to enter a perimeter and a flow area to define the dimensions for a duct of any other shape.

Leakage: Leakage of air between the duct and the surrounding zone may be described in terms of a leakage rate (per unit surface area of duct segment) at a given pressure or by use of the leakage classification. If you enter a leakage rate and a pressure difference, the value for the leakage class is automatically calculated by ContamW and placed in the "Leakage Class" field. Similarly, if you input a leakage class, a default value of 250 Pa is used for the pressure difference, and a leakage rate will be calculated based on the pressure difference and the leakage class you entered. See [ASHRAE 2005 Chapter 35]

Duct Flow Element Types

This is a list of available duct flow elements or mathematical models that provide the relationship between airflow and pressure difference for duct segments. See Airflow Analysis in the Theoretical Background section of the manual for mathematical details.

Darcy-Colebrook Model

The most common model to use for CONTAM duct systems is the Darcy-Weisbach relation and Colebrook's natural roughness function.

Darcy-Colebrook Model

Powerlaw Models

Three versions of the powerlaw model are included:

1. Orifice resistance model
2. Resistance: F = C(ΔP)^n (mass flow)
3. Resistance: Q = C(ΔP)^n (volume flow)

Forced Flow Models

Three types of forced flow models are included:

1. Fan - performance curve
2. Constant mass flow
3. Constant volume flow

Backdraft Damper Models

The Backdraft Damper models make it possible to model a feature (e.g. a smoke control damper) that has different flow resistances depending on the direction of the pressure drop, with greatly reduced (or zero) flow in one direction.

1. Backdraft Damper: F = C(ΔP)^n (mass flow)
2. Backdraft Damper: Q = C(ΔP)^n (volume flow)

Cubic Spline Flow Models

These models provide the ability to create user-defined duct flow elements based on a set of data points to which a cubic spline curve is fit. There are four versions that relate either mass flow or volume flow to pressure difference across the flow element (or vice versa). The cubic spline curve fit insures that the relationship between flow and pressure is differentiable as required by the CONTAM airflow solver.

1. F vs P: *Mass flow rate* as a function of *Pressure difference*
2. Q vs P: *Volume flow rate* as a function of *Pressure difference*
3. P vs F: *Pressure difference* as a function of *Mass flow rate*
4. P vs Q: *Pressure difference* as a function of *Volume flow rate*

Darcy-Colebrook Model

This model uses the Darcy-Weisbach relation and Colebrook's natural roughness function [ASHRAE 2005 p 35.7].

Name: Enter a unique name you want to use to identify the duct flow element. The element will be saved within the current project and can be associated with multiple duct segments.

Description: Field for entering a more detailed description of the specific duct flow element.

Roughness: The flow resistance due to friction is calculated from Colebrook's function and the roughness factor [ASHRAE 2005 p 35.7]. Some typical values are given here.

Smooth	0.03 mm	0.0001 ft
Medium Smooth	0.09 mm	0.0003 ft
Average	0.15 mm	0.0005 ft
Medium Rough	0.90 mm	0.0030 ft
Rough	3.00 mm	0.0100 ft

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Powerlaw Model: Orifice Area

This airflow element allows you to describe the airflow through an orifice, and ContamW converts it to a powerlaw relationship.

Name: Enter a unique name you want to use to identify the duct flow element. The element will be saved within the current project and can be associated with multiple duct segments.

Description: Field for entering a more detailed description of the specific duct flow element.

Flow Exponent (n): Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Discharge Coefficient (C): 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.

Hydraulic Diameter: The hydraulic diameter is equal to (4 · Area / Perimeter). For square openings this equals the square root of the area, and for long thin openings it is two times the width.

Reynolds Number: The transition from laminar flow to turbulent flow occurs over a very broad range of Reynolds numbers with the flow being fully laminar approximately below 100.

Note: The hydraulic diameter and Reynolds number have little impact on the calculations. Generally you should use the default values except for special circumstances where they need to be modified. The values above describe the flow characteristics of an orifice in typical operation. At extremely low pressure drops the use of the powerlaw model leads to a division by zero

during the network solution process. ContamX avoids this problem by changing to a linear model in this region. The model is based conceptually on the flow changing from turbulent to laminar at very low pressures. The Hydraulic diameter and Reynolds number are used to determine a point where the model changes from the powerlaw to linear.

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Powerlaw Model: F=C(ΔP)^n

This airflow element allows you to directly enter the coefficients C and n for the mass flow version of the powerlaw model.

Name: Enter a unique name you want to use to identify the duct flow element. The element will be saved within the current project and can be associated with multiple duct segments.

Description: Field for entering a more detailed description of the specific duct flow element.

Flow Coefficient (C): The coefficients may only be expressed in SI units due to the conversion method used. Use the following conversion to convert from IP units to SI units.

$$\text{To convert from units of } \frac{lbm/s}{in\,H_2O^n} \text{ to SI units of } \frac{kg/s}{Pa^n} \text{ multiply by } \frac{0.4536}{249^n}$$

Flow Exponent (n): Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Powerlaw Model: Q=C(ΔP)^n

This airflow element allows you to directly enter the coefficients C and n for the volumetric flow version of the powerlaw model.

Name: This is the name you give to this airflow element. This name must be unique within a project.

Description: Field for entering a more detailed description of the specific duct flow element.

Flow Coefficient (C): The coefficients may only be expressed in SI units due to the conversion method used. Use the following conversion to convert from IP units to SI units.

$$\text{To convert from units of } \frac{cfm}{in\,H_2O^n} \text{ to SI units of } \frac{m^3/s}{Pa^n} \text{ multiply by } \frac{1}{2119\times 249^n}$$

Flow Exponent (n): Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Fan Model: Performance Curve

This airflow element allows you to create a fan based on a fan performance curve. You provide ContamW with a set of pressure rise and airflow rate data points, and ContamW performs a polynomial curve fit to the data point.

Name: Enter a unique name you want to use to identify the duct flow element. The element will be saved within the current project and can be associated with multiple duct segments.

Fan off conditions: These values specifies how the fan is modeled when the speed of the fan is insufficient to provide a flow.

Cut-off Ratio: The *fan speed ratio* is the actual speed (RPM) of the fan divided by the rated speed of the fan (fan is defined for a speed ratio of 1.0). When the fan speed ratio is below the specified *cut-off ratio* the fan is modeled as a simple orifice based on the "equivalent orifice" defined for this fan element. During a simulation, ContamX obtains the actual ratio from the control value being applied (in the form of a schedule or control signal) to the airflow path with which this fan element is associated and compares it to the *cut-off ratio*. See Chapter 18 Fans in [ASHRAE 2004].

> **Equivalent Orifice:** The size of the orifice used to model the fan opening when the schedule or control value drops below the specified cut-off ratio.

Description: Field for entering a more detailed description of the specific airflow element.

Fan Performance Data: Click the "Edit Fan Curve Data" to input/modify the fan curve for this duct flow element. A detailed explanation of this data is given in the Fan Performance Data section under Working with Airflow Paths.

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Fan Model: Constant Mass Flow

This airflow element allows you to describe the airflow through a duct as a constant mass airflow rate.

Name: Enter a unique name you want to use to identify the duct flow element. The element will be saved within the current project and can be associated with multiple duct segments.

Description: Field for entering a more detailed description of the specific duct flow element.

Design (max) Flow: Enter the maximum mass flow rate. You can use the duct segment schedule to modify this value.

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Fan Model: Constant Volume Flow

This duct flow element allows you to describe the airflow through a duct as a constant volumetric flow rate. The design flow rate you specify is in terms of standard air. CONTAM uses a value of 1.204 kg/m^3 based on conditions specified by ASHRAE for dry air: 101.325 kPa and 20ºC [Chapter 18 of ASHRAE 2004]. Therefore, if actual conditions during simulation do not match these standard conditions, the results will differ from specified flow due to differences in air density.

Name: Enter a unique name you want to use to identify the duct flow element. The element will be saved within the current project and can be associated with multiple duct segments.

Description: Field for entering a more detailed description of the specific duct flow element.

Design (max) Flow: Enter the maximum volume flow rate. You can use the duct segment schedule to modify this value.

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Backdraft Damper: Q=C(ΔP)^n

This is the volumetric flow version of the Backdraft Damper duct flow element. The Backdraft Damper models make it possible to model a feature (e.g. a smoke control damper) that has different flow resistances depending on the direction of the pressure drop, with greatly reduced (or zero) flow in one direction.

Name: Enter a unique name you want to use to identify the duct flow element. The element will be saved within the current project and can be associated with multiple duct segments.

Description: Field for entering a more detailed description of the specific duct flow element.

Pressure Difference: Enter a flow coefficient and exponent for both the positive and negative flow directions of the duct segment. **ΔP>0** refers to the pressure difference across the flow element that would lead to an airflow in the positive flow direction (See Duct Segment Properties) and **ΔP<0** would lead to a flow in the opposite direction.

Flow Coefficients: The coefficients may only be expressed in SI units due to the conversion method used. Use the following conversion to convert from IP units to SI units.

$$\text{To convert from units of } \frac{cfm}{in\,H_2O^n} \text{ to SI units of } \frac{m^3/s}{Pa^n} \text{ multiply by } \frac{1}{2119\times 249^n}$$

Flow Exponents: Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Backdraft Damper: F=C(ΔP)^n

This is the mass flow version of the Backdraft Damper duct flow element. The Backdraft Damper models make it possible to model a feature (e.g. a smoke control damper) that has different flow resistances depending on the direction of the pressure drop, with greatly reduced (or zero) flow in one direction.

Name: Enter a unique name you want to use to identify the duct flow element. The element will be saved within the current project and can be associated with multiple duct segments.

Description: Field for entering a more detailed description of the specific duct flow element.

Pressure Difference: Enter a flow coefficient and exponent for both the positive and negative flow directions of the duct segment. **ΔP>0** refers to the pressure difference across the flow element that would lead to an airflow in the positive flow direction (See Duct Segment Properties) and **ΔP<0** would lead to a flow in the opposite direction.

Flow Coefficients: The coefficients may only be expressed in SI units due to the conversion method used. Use the following conversion to convert from the IP units to SI units.

To convert from units of $\frac{lbm/s}{in\,H_2O^n}$ to SI units of $\frac{kg/s}{Pa^n}$ multiply by $\frac{0.4536}{249^n}$

Flow Exponents: Flow exponents vary from 0.5 for large openings where the flow is dominated by dynamic effects, and 1.0 for narrow openings dominated by viscous effects. Measurements usually indicate a flow exponent of 0.6 to 0.7 for typical infiltration openings.

Shape Size and Leakage: You must enter data to physically describe each duct airflow element. You input these values on the Shape Size and Leakage property page associated with each duct airflow element.

Duct Junction and Terminal Properties

This section provides detailed descriptions of the specific duct junction and terminal properties. The following sections are the context-sensitive help topics that you can access by pressing **F1** when working with property pages of the "Duct Junction Properties" property sheet.

Duct Junction and Terminal - Junction Properties

Junction Number: This number is automatically generated by ContamW for identification purposes and is unique to each junction. ContamW will renumber junctions and terminals when saving a project if you have added or removed any junctions or terminals. These icons are numbered starting at the top level in the upper left hand corner of the SketchPad moving left to right and down the SketchPad then proceeding down through each level in the same manner.

Relative Elevation & Temperature: The elevation and temperature of the junction are used to determine how the duct flow responds to and influences the building stack effect. Enter the height of the midpoint of the junction or terminal point relative to the current building level. Enter the temperature you wish the junction or terminal to use during simulation.

Note that you can have ContamX calculate duct temperatures when performing simulations using the 1D Duct Model with the Short Time Step Method. See Contaminant Numerics Properties in the Working with Simulations section.

Junction: If you are creating a junction that connects two duct segments, then select the type of connection to create here. Items will be enabled/disabled depending on the valid connection types.

> **Define Downward Duct:** You define vertical duct segments as downward running ducts accessed via the duct junction at the top of the downward segment. Press the "Define Downward Duct" button to define vertical segments. The SketchPad duct icons that are connected to the defined vertical duct will change to indicate the type of vertical connections associated with the junction, i.e., connected downward, upward or both (see Junction and Terminal Icons).

Terminal: If you are creating a terminal at the end of a duct segment, then select the type of connection to create here. Items will be enabled/disabled depending on the valid connection types.

Location: Enter the absolute X and Y coordinates and units for the selected junction. The Z coordinate will be taken as the *Relative Height* of the junction with respect to the building level on which the junction is located.

These values are required for *terminals* located in the Ambient zone when using a *Wind Pressure and Contaminant (WPC)* file (see Working with WPC Files) or for *junctions* and *terminals* located within a *1D convection/diffusion zone* (see 1D Zone Data). These coordinates will be verified by ContamW against a WPC file or the axis of the 1D zone in which the junction or terminal is located. This verification will be performed automatically when you select either the **Run Simulation** or **Create a ContamX Input File** command from the **Simulation** menu.

Duct Junction and Terminal - Wind Pressure Properties

This set of properties is available only for those *terminals* that are located in the ambient zone and thus potentially subject to wind effects.

Wind Pressure Option: There are three wind pressure options for duct terminals located in an ambient zone:

1. No wind pressure
2. Constant pressure
3. Pressure dependent on wind speed and direction

When you select one of the above options, the following data entry options that correspond to the wind pressure option that you selected will be made available for your input.

Constant Pressure Data: Enter a constant wind pressure (wind speed and direction independent).

Variable Pressure Data: Variable wind pressures are computed from the product of the following three parameters:

1. The dynamic pressure of the wind at some reference height.
   ContamX will determine this pressure based on either the steady state wind data (See Wind Properties) or the transient wind data from a weather file (See Defining Transient Weather and Wind).
2. A coefficient accounting for local terrain effects (Wind Pressure Modifier).

3. A pressure coefficient accounting for relative wind direction (Wind Pressure Profile).

   Wind Pressure Modifier: CONTAM uses this value to account for differences between wind velocity profiles at the building site and that at a measurement location, e.g. an airport weather station (See Working with Weather and Wind). CONTAM only requires this value if you are implementing Variable Wind Pressure for an airflow path. ContamW will provide a default value for this based on the data entered via the Weather and Wind Parameters Property Page or you can override the default value for a particular airflow path.

   Wall Azimuth Angle: Enter the direction the wall faces (clockwise from north).

   Profile: The wind pressure profile accounts for the wind direction effects. You may select a previously defined wind pressure profile from those contained in the Name list. Click the "New Profile" button to define a new wind pressure profile. To view or modify wind pressure profile data, click the "Edit Profile" button. After you press the "New Profile" or "Edit Profile" button, the Wind Pressure Profile page will be displayed with a graphical representation of the profile.

Duct Junction and Terminal - Terminal Properties

Prior to CONTAM 2.4, duct terminals merely provided connectivity of the duct network to zones and did not have user-defined properties associated with them. Beginning with CONTAM 2.4, duct terminals have been revised to be more like duct flow segments, having geometric properties, loss coefficients and flow balancing capabilities.

Terminal flow is determined by the *effective loss coefficient* (C_e) of the terminal. C_e is determined during simulation by the combination of the Terminal Loss Coefficient (C_t), Balance Coefficient (C_b) and schedule value or control signal (S_c). As implied here, terminals can now be acted upon via the control network or scheduled to allow C_e to range between ($C_t + C_b$) for a control signal of 1.0 and a very large loss coefficient that essentially reduces the flow to zero for a control signal of 0.0. (See Ducts in the Airflow Analysis section of the Theoretical Background).

You can use the terminal properties to define the loss characteristics of a duct terminal to be used during simulation; to provide geometric data that will enable ContamW to provide you with more meaningful flow velocities at the terminal; and to define duct balancing properties that CONTAM can use to establish user-defined design airflow rates at the duct terminals. After setting the design conditions below, you can have CONTAM perform the duct balancing by selecting the Duct Balance airflow simulation method of the Run Control Properties.

Junction Number: This number is automatically generated by ContamW for identification purposes and is unique to each junction. ContamW will renumber junctions and terminals when saving a project if you have added or removed any junctions or terminals. These icons are numbered starting at the top level in the upper left hand corner of the SketchPad moving left to right and down the SketchPad then proceeding down through each level in the same manner.

Terminal Data: These properties are the characteristic data that describe the terminal itself regardless of whether it is balanced or not.

   Terminal Loss Coefficient: Characteristic loss coefficient of the terminal. The default value is 0.125. See chapter 35 of [ASHRAE 2005] and the Duct Fitting Database [ASHRAE 2002] for data on dynamic losses in duct fittings.

NOTE: Earlier versions of CONTAM (version 2.2 and earlier) did not implement loss coefficients at the terminals. When updating these earlier versions of CONTAM PRJ files, terminal loss coefficients will be applied to all terminals. For those terminals whose adjacent duct segments had loss coefficients associated with them, half of the segment loss coefficient will be applied to the terminal and half to the adjacent segment. All other terminals will be provided with a minimum allowable value of 0.001, thus effecting a very small loss at the terminal to provide consistency with terminal flow results prior to the conversion.

Free Face Area: This is the total minimum area of the openings in the air outlet or inlet through which air can pass. This area will be used to calculate the velocity from the airflow rate at the terminal when displaying simulation results on the SketchPad.

Duct Area: The area of the duct to which the terminal is connected. Typically, this would be the same area as the duct segment just downstream/upstream from the supply/return air terminal. This value is used during simulation to determine the airflow rate through the terminal.

Balancing Data: These properties are used for the automated duct balancing capabilities implemented in CONTAM version 2.4. This feature will enable you to select a set of duct terminals for which you provide design airflow rates and have ContamX calculate loss coefficients that will enable the associated supply fan to provide the design flow rate.

Balance Terminal: Check this box if you would like ContamX to include this terminal in the duct balancing procedure.

Design Flow Rate: Provide the design airflow rate for which you want ContamX to determine a *Balance Loss Coefficient*. Specify positive values for supply flow terminals, i.e. airflow from the duct system to the zone in which the terminal is located, and negative values for return air terminals.

Max Balance Coefficient: Provide a maximum value for the Balance Loss Coefficient below. This will be used to provide warning information if the *Balance Loss Coefficient* determined by the automated balancing procedure of ContamX exceeds this value.

Balance Loss Coefficient: This is the loss coefficient determined via the automated balancing procedure of ContamX. You can also modify it "manually." As noted above, this value will be used to determine the effective loss coefficient (C_e) for the terminal.

Filter and Schedule

Junctions and terminals differ with respect to Filters and Schedules. Junctions can not have filters associated with them and only the temperature of the junction can be scheduled or controlled. Beginning with CONTAM version 2.4, terminals can be both filtered and scheduled. The terminal temperature and loss coefficient can be scheduled (or controlled), although only one type of schedule can be applied at a time.

Filter: You may either create, edit or delete the filter associated with this terminal. Creating a new filter or modifying an existing one will display the Filter Properties dialog box. Use this dialog box to create and edit filter elements as needed. For an explanation of contaminant filters see Contaminant Filters in the Using CONTAM section of the manual.

Schedule: As of CONTAM version 2.4, terminals consist of both a duct segment and a terminal junction during simulation. Each of these are schedulable items. You can select to schedule

either the **Loss coefficient** of the segment or the **Temperature** of the junction. The list of schedules will contain those types (unitless or temperature) of schedules that exist in the project.

If there are previously defined schedules within the current project, you may select one from the Name list. Click the "New Schedule" button to define a new schedule. To view or modify an existing schedule click the "Edit Schedule" button. Setting this field to <none> indicates no schedule. (See Working with Schedules)

CVF/DVF Control: Values from either a CVF or DVF file can be used in place of the above Schedule (read about CVF and DVF controls in the Control Element Types section). This enables using these control element types without having to draw/define an associated control network. The associated radio buttons "Use Neither", "Use CVF" and "Use DVF" are used to select the desired control method. Select the heading name from the associated "Name" drop down box that will display the headings of either the CVF or DVF file as per the aforementioned radio button selection.

Control Status: This will display the current method of control: Schedule, direct CVF, direct DVF or Control Network. If an actuator of a control network on the SketchPad is associated with the scheduled item then it will take precedence over the Schedule and CVF/DVF Control. If either the "Use CVF" or "Use DVF" radio button is selected then CVF or DVF will take precedence over an associated schedule.

Balancing Duct Systems

In CONTAM you can balance duct systems yourself by providing balance points, e.g., orifice flow elements, at desired locations, or you can have CONTAM balance the system for you. Versions of CONTAM prior to version 2.4 required you to balance the system manually by adjusting the resistance of duct flow elements or local loss coefficients at desired balancing points. Beginning with version 2.4, CONTAM provides the ability to automatically balance duct systems using CONTAM duct terminals. See Duct Junction and Terminal - Terminal Properties and Working with Simulations.

Automated Duct Balancing Procedure

Automated duct balancing is another mode of simulation in ContamX. When you initiate a balance, ContamW will perform a pre-simulation check on the PRJ file and, if the check is successful, save the PRJ file and execute ContamX. ContamX will perform its own set of pre-balance checks to determine if a valid set of conditions has been defined. If this check is successful, then the automated balancing mode of ContamX will be initiated. If the pre-check of ContamX is unsuccessful, an error message will be displayed indicating the nature of the pre-check failure.

The automated procedure establishes relationships between fan elements and duct terminals by first performing a steady state airflow simulation for the entire building system as defined, i.e., duct balance coefficients having not yet been modified. If there are no errors in the system configuration, it then replaces fan elements with constant flow balancing fans, initializes fan flow rates to the summation of design flows of associated terminals (the minimum flow required for balancing), performs an airflow simulation, and checks for convergence of the duct balance by comparing calculated airflows with design airflows for all balance terminals. If convergence is not met, then the balance coefficients are adjusted (only increased) and the procedure is repeated until either convergence or the maximum number of iterations are performed. Upon

completion of the balancing, ContamX will create a simulation results file (SIM file) that contains the calculated airflow results of the balanced system (see items 0 and 1 below).

The automated balance procedure generates a *duct balance result file* which has the same name as the project file but having the BAL extension. The contents of the balance report file will vary depending on the results of the automated balance procedure (See Results Files in the Working with Results section).

The basic steps to use the duct balancing feature are outlined below:

- Draw duct system – define segments, terminals and fans. Be sure to use separate fan flow elements for each fan, as the balance procedure may need to adjust the flow rates individually to achieve desired results. In order to obtain as realistic and reasonable results as possible, you should provide known loss coefficients at each terminal and junction throughout the system. Otherwise, the balancing procedure might not produce desired static pressures within the system – although design flows will be achieved.
- Designate balancing terminals and set design airflow rates for each.
- Select airflow simulation balance procedure and run simulation.
- Observe feedback from balance procedure. There are six possible results. If the one of the first two results (Success or Non-convergence) is obtained, a simulation results file will be created that contains the airflow rates calculated based on the calculated balance coefficients. If one of the four error results is obtained, then the simulation results are for the pre-balance steady state calculation of the system as defined. The following list provides the six different outcomes of the balance and their associated completion code as it will appear in the first line of the *duct balance results file*.

0. Success: The balance procedure completed successfully. You will be prompted to copy balance coefficients to terminals and balance flow rates to constant flow fan elements when using such elements to model the fan. Once you have responded to the prompts, you can review the results of the balance on the SketchPad. Airflow rates from the terminals will be displayed via the SketchPad results display method. Balance coefficients will not be saved until you save the project, so you can maintain the original pre-balance file by saving the post-balance file under a new name.

Performance curve fan elements are treated differently from constant flow fan elements, because they can not be directly modified to provide the balance flow rates. If using fan curve elements, consult the balance report file for setting the fan speed ratio to provide design flow rates. Use the value reported to provide a speed ratio via a control signal or schedule value to achieve the balance flow with the fan. If a value of 0.0 is reported, then the fan curve is not capable of providing the desired flow via a reasonable fan speed ratio and pressure rise. In this case it is likely that a different fan should be used.

1. Non-convergence: The balance procedure was not able to obtain the desired balance flow rates for all of the terminals within the preset number of iterations. You can choose to accept the calculated balance coefficients and run the balance procedure again with the modified coefficients as initial values. The balance report file will provide a relative flow coefficient equal to the calculated flow rate divided by the design flow rate.

2. Error: No terminals to balance: There are no terminals having the *Balance Terminal* option selected. Be sure to check the *Balance Terminal* check box to designate terminals to be balanced.

3. Error: Invalid flow direction(s): This error occurs when the balance terminals associated with a single fan have opposing design flow directions. Be sure the design flow direction of all terminals matches that of the associated fan: positive design flow rates are out of the terminal into the zone in which the terminal is located, and negative design flow rates are into the terminal from the zone. The fan flow direction is user-selectable via the duct segment icon of the associated fan.

4. Error: Failed to terminate link(s) at a fan: This error occurs when one or more balance terminals can not be traced back through the duct system (links) to a forced flow element (fan) within the duct system.

5. Error: Fan system includes supply and return terminal: This error occurs when a given fan has balance terminals both upstream and downstream of the fan. Be sure to only attempt to balance terminals on either the upstream or downstream side of a fan.

DUCT BALANCING NOTES:

- Duct balancing is performed under consideration of the entire building system. When the airflow calculation is performed during each iteration of the balancing procedure, the resistances between zones are included. Interzone resistances that are relatively high can impact the balancing results.
- During duct balancing, all junction temperatures are set to 20 ºC. Ambient conditions are established by the steady state weather data with the exception that wind is set to 0 m/s.

3.8 WORKING WITH CONTROLS

Controls allow the characteristics of one building element to be modified during transient simulations in response to a change in characteristics of another building element. You can also use controls to simply monitor building performance by polling building components for values and logging them to a control log file during a simulation.

Controls include sensors, links, modifiers and actuators. Sensors are control elements that are used to obtain a value, calculated during simulation, of a building component and then provide it to a control network. You can use sensors to obtain contaminant concentrations and temperatures of zones, duct junctions and duct terminals and to obtain pressure differences and airflow rates of airflow paths, duct segments and duct terminals. You can use actuators to control schedulable items including airflow paths, duct segments, junctions and terminals, simple air-handling systems, and contaminant source/sinks. Note that actuator input to a controlled building component overrides schedules associated with the component.

The connection between sensors and other nodes in a control network are made up of control links. Control modifiers are nodes within the control network that are used to change or modify their input signals. Control actuators are used to modify various characteristics of building components based on control signals obtained from sensors and even modified by signal modifiers. For example, a sensor can be used to obtain a contaminant concentration within a zone, and a proportional control actuator can be used to adjust supply airflow through a supply airflow path into the zone based on the sensed concentration.

3.8.1 Control Elements

A control network can consist of many different control nodes, each of which is defined by a control element. There are several different types of control elements provided, such as a report element, that can be used to convert dimensionless sensor values to engineering units; mathematical elements to perform simple mathematical and logical operations on input signals; and classical control elements such as proportional/integral, band and limit controls. There is a phantom control element that can be used to reference an existing control node, which is convenient for linking nodes that may be on different levels of a building or simply too far apart to conveniently draw a physical link. Each of these element types is described in detail in the Control Element Types section. While every node in the control network must refer to a single control element, multiple nodes can refer to the same control element using the phantom node/control element.

Collections of control elements can be grouped into Control Super Elements, which can be stored in CONTAM Library files and shared within and between projects. Super elements can reduce the amount of drawing of repetitive control logic and enable the creation of custom reusable sensors and control algorithms.

3.8.2 Drawing Controls

You draw controls using the links drawing tools previously described in the "Working with the SketchPad" section. When you finalize the drawing of a set of control links, the appropriate icons will be placed upon the SketchPad depending on where the link originates and where it terminates. For example, if you begin drawing a control link directly on a zone icon, a sensor

icon will be placed adjacent to the zone icon, and if you terminate a control link on an airflow path, an actuator icon will be placed adjacent to the airflow path icon (See Figure below).

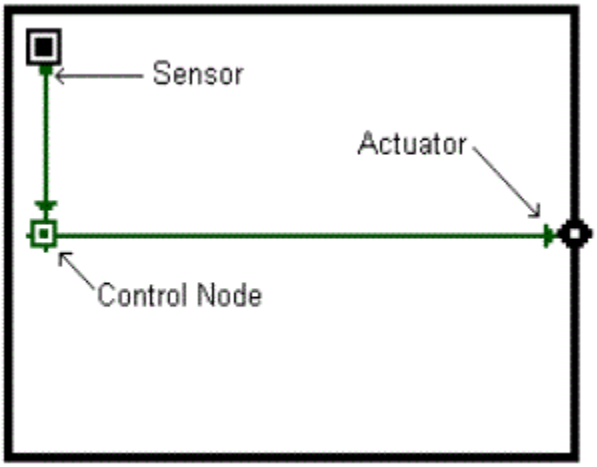


Figure – Control link icons

If you terminate a control link on a blank cell, a control node icon will be placed within that cell. You may not originate and terminate a link on a non-control node icon (e.g. zone, flow path or occupant) without an intermediate control node icon (See Figure below). This is due to the requirement that the input from an actuator to a non-control node icon must be in the form of a multiplier ranging from 0.0 to 1.0.

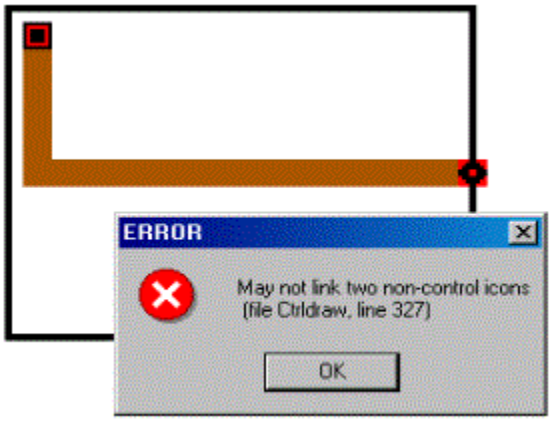


Figure – Invalid control link

Control node icons can have up to four links into or out of them depending on the type of control node you are defining. If a control node icon is not yet defined, ContamW will only allow you to draw at most two inputs (one vertical and one horizontal) into the node and at most four outlets from the node. However, if the control node icon is already defined, ContamW will only allow the correct set of links to be defined into the icon depending on the *control element type* (See Control Element Types) of the control node. This is due to the input signal convention of control

nodes that distinguishes two different inputs signals for certain element types, e.g. limit controls, or to establish the order of operators for non-commutative mathematical operations, e.g. subtraction and division. As shown in the Figure below, *Input Signal 1* enters either from the left or right and *Input Signal 2* enters from either the top or bottom of the node. Details on input signals are provided for each element type where required in the description of the specific elements.

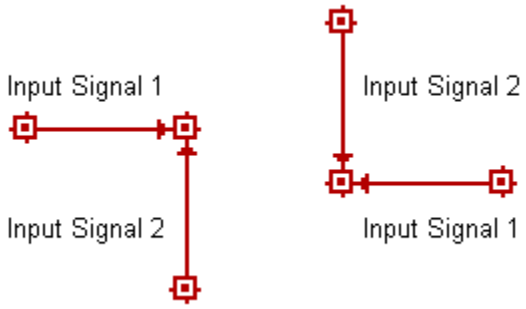


Figure – Control node input signal convention

3.8.3 Deleting Controls

Deleting currently defined controls is a two-step process and can only be performed when ContamW is in the *normal* mode, i.e., not viewing SketchPad results. That is, if you want to delete a control link, sensor, actuator or node, you must first *undefine* the item. To undefine a control you must highlight a control node or link and press the **Delete** key or select Delete from the Edit menu. This will highlight either a single node (if no output signals) or a link segment and prompt you to confirm the undefine operation by displaying a message box. Once you undefine a control node or link segment, it will again be displayed in red. To delete the control node or link you highlight any portion of the undefined node or link segment and perform the deletion. The entire portion to be deleted will be highlighted, and you will be asked to confirm the deletion. If you try to delete a node that is referenced by a phantom control element, you will be warned. Deleting such elements prior to deleting those phantom nodes that reference them can cause problems.

3.8.4 Hiding Controls

If there are controls in your project, you can hide them on the SketchPad. Use the **View →Control Links** menu command to toggle the display of the control network on and off.

3.8.5 Modifying Controls

To modify the parameters of a defined control node, you use the icon definition procedure (See Defining Building Component Icons) to display its properties and make the desired changes. You can not move and copy controls. To do this you must delete and redraw your control network, therefore careful planning is advised in laying out your control networks.

3.8.6 Defining Controls

You must define all *control nodes* and *sensor* icons in order to perform a simulation. Control links will be defined automatically upon definition of the node or sensor from which they emanate. Generally, you should define control nodes in the direction of information flow

indicated by the link arrows. Define *control nodes* and *sensors* using the icon definition procedure (See Defining Building Component Icons) to display and edit their properties.

Sensors

When you double-click an undefined sensor icon, the Control Sensor definition dialog box will appear. The parameters will either be those for a *zone sensor* or *flow path sensor* depending on the building component with which the sensor is associated.

As noted in the Drawing Controls section above, sensor icons can be placed automatically when you begin drawing a control link on a sensible icon, e.g., zone or flow path. With CONTAM version 2.4, you can now define zone sensors within any blank cell inside of a zone. This increases the previous limit of four zone sensors per zone and is done by selecting the "Sensor" control element type when defining a control node as presented in the following sub-section.

Control Nodes

Double-clicking on an undefined control node icon will display the Control Element Types dialog box from which you select the type of control element you wish to define. Control elements can be categorized by the number of inputs that can be associated with them: unary, binary and multiple. Therefore, when the Control Element Types dialog is displayed, various types will be enabled/disabled for selection depending on the number of inputs that you have drawn into the currently selected node.

Once you select the type of element you want to create, you assign values to the parameters of the control elements. All control elements have an optional *Name* parameter with the exception of the phantom for which the *Name* parameter will be used to reference the name of another existing element. The name parameter is only required if you want to reference the element with a phantom node.

Each element also has an optional *Description* field, which is filled in with a default value when you create an element, but you may change the description. The description will be displayed in the status bar when you highlight the control node on the SketchPad. Many of the elements do not have any parameters other than the *Name* and *Description*, because they simply perform a predetermined operation on the input(s) to the node, e.g., the "+ Add" element simply takes two inputs, adds them and provides the results as an output signal. Other control elements have additional parameters required to define their behavior, e.g., the "Constant" element has a value parameter use to specify the constant value of the output signal from the control node.

3.8.7 Control Element Types

There are a number of control element types that you can associate with an undefined control node. Detailed descriptions and properties of these control elements are provided in the following sections.

Sensor

There are two types of sensors that can be created depending on the type of icon with which the sensor is associated: *Zone or Junction Sensor* and *Path or Duct Sensor*. The Sensor definition dialog box will provide you with the properties available for the type of sensor being created.

Zone or Junction Sensor

Sensed Value: Select the property of the zone or junction that you would like the sensor to provide as output: ***Temperature***, ***Contaminant mass fraction***, ***Gage pressure*** or ***Occupancy***.

Path or Duct Sensor

Sensed Value: Select the property of the path or duct that you would like the sensor to provide as output: ***Flow rate*** through the path or ***Pressure drop*** across the path.

Sensor Output: Sensor output defaults to the base units of CONTAM, but you can modify this by changing the *Gain* and *Offset* values provided. However, the Report a Value control element (described below) automatically provides values to convert from the base units to the default units of your project or of the particular contaminant which you are sensing.

$$Out = (Sensor\ Challenge\ Value - Offset) * Gain$$

Gain: a multiplier used to convert the *Sensor Challenge Value* to the *Out* value

Offset: an offset value used to convert the *Sensed Value* to the *Output* value

Time Constant: sensor response time constant in units of seconds. ContamX will use this to calculate a delay factor.

$$\text{Delay} = e^{-\Delta t/\tau}$$

The sensor output is calculated from the converted *Sensor Challenge Value* for the current time step t (Out), the output from the previous time step t-Δt, and the *Delay* value as follows:

$$\text{Output}(t) = Out + [\text{Output}(t-\Delta t) - Out] * Delay$$

Location: These parameters only apply to *Mass fraction sensors* associated with 1D zones. Select whether you want the sensor to provide the **Zone Average** concentration of a 1D zone or the value within a single **1D Cell** of a 1D zone. The 1D cell option is only effective when performing a simulation using the *short time step* method otherwise the sensor will provide a value based on the zone average.

X, Y and **Rel Elevation** (Z) are the coordinates and units for the selected sensor. X and Y are absolute coordinates and Rel Elevation is relative to the level on which the sensor is located. These values are required for any sensor that is located within a cell of a 1D *convection/diffusion zone* (see 1D Zone Data). If the sensor is to be located within a particular cell of a 1D zone, then these coordinates will be verified by ContamW against the axis of the 1D zone in which it is located. This verification will be performed automatically when you select either the **Run Simulation** or **Create a ContamX Input File** command from the **Simulation** menu.

Description: Use this to describe the sensor in detail. This description will be displayed in the status bar when you highlight the sensor icon. The description defaults to either "zone sensor" or "path sensor."

Phantom control

Use this control element to reference a named control element that already exists elsewhere within your project. Phantom nodes are useful when you need a control signal from a node that is located remotely upon the SketchPad or when you are implementing one of the multiple-input elements. This is useful in referencing a node on another level of a building, because ContamW does not allow you to draw links between building levels. This can also be used to "cascade" inputs into the multiple-input *sum* and *average* controls when you want to sum or average more than three inputs or remotely located, named inputs.

Schedule

Use this control element to apply a schedule via the control network. A schedule can be applied to airflow paths, simple air handling systems, inlets and outlets of simple air handling systems and source/sinks. Control schedules will override schedules that are defined as a parameter of an element. You can select an existing schedule or create a new one (See Creating Schedules).

Constant

Use this control element to define a simple constant. The constant is useful as input to other control elements such as the *mathematical*, *limit* and *switch* controls.

Modifier

Use this control element to modify an input signal by providing a Gain and Offset parameter. The input signal will be modified according to the following equation.

output = (*input* – Offset) * Gain

The parameters for the Modifier control are:

Gain - a multiplier used to convert the *input* signal

Offset - an offset value used to convert the *input* signal

Hysteresis

Use this control element to simulate hysteresis of a control actuator. Hysteresis is characterized by the Slack parameter where Slack is the fraction of the input signal's range over which the output signal remains constant when the input signal changes direction. The relationship of the output signal to the input signal is shown in the following figure. This plot shows the signal increasing from 0 to 1 and back down to 0 again. The increasing and decreasing signals are characterized by the following equations respectively.

Increasing: *output* = maximum(0.0, *input* * Slope + Int)

Decreasing: *output* = minimum(1.0, *input* * Slope)

Where

Slope = 1 / (1.0 – Slack)

Int = *output* – Slope * *input*

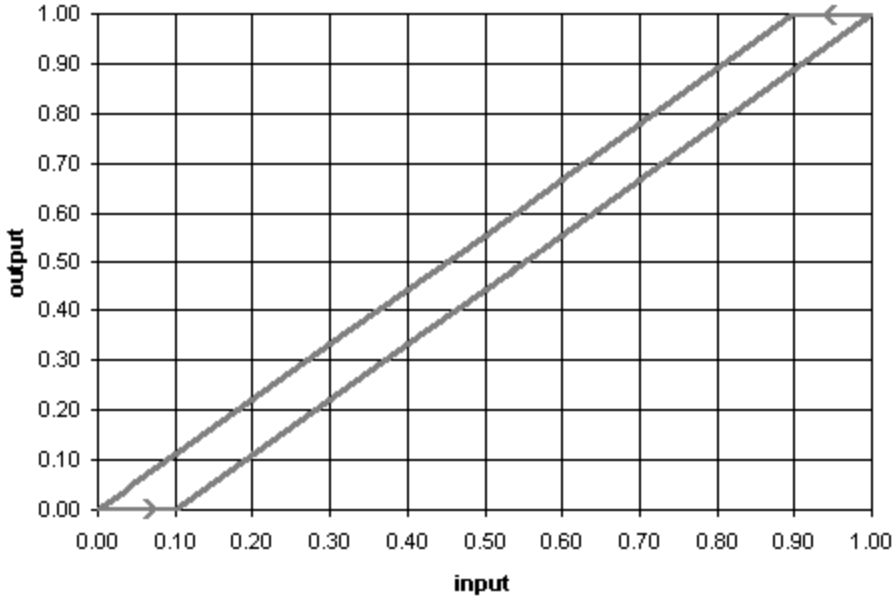


Figure – Plot showing hysteresis curve of Hysteresis control element

Report a value

Use this control element to report values to a file created by ContamW for the specific purpose of logging these reported values for each *listing time step* of a transient simulation (See Output Properties). If you perform a transient simulation and implement *report control elements*, ContamW will create a file in the same directory as your project file with the name of your project file and the .LOG extension appended. For example if your project is *MyProj.PRJ*, ContamW will create the file *MyProj.LOG*. You can also use the Report control element to provide automatic unit conversions of sensor data from the dimensionless units of a sensor (e.g. K or kg/kg) to engineering units associated with the type of sensed data (e.g. ºC or ppm(v)).

The value output from the report element will be determined according to the following equation:

output = (*input* – Offset) * Gain

The parameters for the Report control are:

Gain: a multiplier used to convert the *input* signal

Offset: an offset value used to convert the *input* signal

Header: the header that will appear in the log file for this control element

Units: units that will appear below the header in the log file for this control element

The report control will also provide the default conversion values, Gain, Offset and Units if ContamW finds a sensor "down-link" in the control network that has not yet been converted from its dimensionless form.

Signal split

Use this to split a signal, so the same signal can be directly linked to multiple nodes in the control network.

Logical

Use these controls to provide logical operations including:

Binary – convert input signal to binary 0 or 1,

NOT – negate single input value and provide binary output 0 or 1,

AND – perform AND operation on two input values and provide binary output 0 or 1.

output = 1 IF (*input1* > 0) AND (*input2* > 0)

output = 0 otherwise

OR – perform OR operation on two input values and provide binary output 0 or 1.

output = 1 IF (*input1* > 0) OR (*input2* > 0)

output = 0 otherwise

XOR – perform XOR operation on two input values and provide binary output 0 or 1.

output = 1 IF (*input1* > 0) AND (*input2* <= 0)

output = 1 IF (*input2* > 0) AND (*input1* <= 0)

output = 0 otherwise

Mathematical

Use these controls to provide mathematical operations including:

Absolute value – provide absolute value of input signal

Add, Subtract, Multiply, and Divide – perform operation on two input signals and provide result as output.

Note the order of operation is important for Subtract and Divide.
Subtract: *output = input1 – input2*
Divide: *output = input1 / input2*

Sum and Average – Sum or average multiple input signals and provide result as output signal. Three signals can be directly input to using links drawn into the control node icon, and more can be cascaded through the use of *phantom* nodes.

EXP(x) – raise e to the power of x.

LN(x) – calculate natural logarithm of x.

LOG10(x) – calculate log base 10 of x.

POW(x,y) – raise x to the power of y where x is *input1* and y is *input2*.

SQRT(x) – calculate the square root of x.

Polynomial(x) – calculate polynomial of x based on user-defined polynomial coefficients.

SIN(x) – calculate sine of x.

COS(x) – calculate cosine of x.

TAN(x) – calculate tangent of x.

MOD(x,y) – calculate the floating point remainder of x divided by y where x is *input1* and y is *input2*.

CEIL(x) – calculate the smallest integer that is greater than or equal to x.

FLOOR(x) – calculate the largest integer that is less than or equal to x.

Limit switches

Limit switches provide the ability to set an output signal to either on or off (0 or 1) based upon the difference between the two input signals.

Lower limit switch

output = 1 IF (*input1* < Limit) [Limit = *input2*]

output = 0 otherwise.

Upper limit switch

output = 1 IF (*input1* > Limit) [Limit = *input2*]

output = 0 otherwise

The band switches incorporate the control parameter, **Band width**, to provide a *dead band* for the on/off control.

Lower band switch

output = 1 when *input1* rises above *input2* + Band width

output = 0 when *input1* < *input2*

input2 is the lower limit of the band

Upper band switch

output = 1 when *input1* falls below *input2* – Band width

output = 0 when *input1* > *input2*

input2 is the upper limit of the band

Limit controls

Limit controls provide the ability to set an output signal to a value ranging from 0.0 to 1.0 based upon the difference between the two input signals. These would typically be used to provide an *error signal* to a proportional controller.

Lower limit control

output = *input2* - *input1*

Upper limit control

output = *input1* - *input2*

The *upper and lower limit controls* can be used to prepare an error signal for the *proportional controls*. If the input signal to an *upper limit control* is greater than the limit value, a positive output results; e.g., if a contaminant concentration is too high, the positive signal could activate an airflow device. If the input signal to a *lower limit control* is lower than the limit value, a positive output results; e.g., if the air flow through an opening is too low, a positive signal could activate a fan to provide the required flow.

Proportional and Proportional–Integral

Proportional control

This is a model of a simple proportional controller where the input signal is an error signal that is typically a sensed value minus a set-point value which is obtained by using other control functions, e.g. the *constant control*. The input signal is multiplied by the proportionality constant parameter of the control, **Kp**. The output signal is limited to values between 0 and 1, so you should set the proportionality constant accordingly. The output signal could then be modified as required using other control elements.

$$output = input \cdot \mathrm{Kp}$$

Proportional-Integral control

This is a model of a simple P-I controller where the input signal is an error signal. The output signal is limited to values between 0 and 1 and is modified by the control parameters **Kp** and **Ki** according to the following equation.

$$output = output^* + \mathrm{Kp} \cdot (input - input^*) + \mathrm{Ki} \cdot (input + input^*)$$

where $output^*$ and $input^*$ are the values of *output* and *input* at the previous time step. The Kp and Ki factors must be tuned for the specific problem and time step. This is similar to an "incremental" P-I algorithm described in equations (17-37) and (18-3) of [Stoecker and Stoecker 1989].

Scheduled Delay

This control element provides the ability to simulate time delays associated with the ramping up/down of system components changing between states, e.g., the spin down of a fan or the opening/closing of a damper. The *scheduled delay* element allows you to define a schedule according to which the change of state occurs. The *output* will change according to this schedule when the *input* changes.

Node Name: This is an optional name you can provide for this control node. This name can be used to reference this node with a *Phantom control node*.

Schedule - Signal Increasing: Select/enter a day schedule that characterizes the delay in an increasing signal. The schedule must be *trapezoidal* beginning with a value of 0.0 at time 00:00:00 and increase to a value of 1.0 before 24:00:00. Only one increasing time period per schedule will be allowed.

Schedule - Signal Decreasing: Select/enter a day schedule that characterizes the delay in a decreasing signal. The schedule must be *trapezoidal* beginning with a value of 1.0 at time 00:00:00 and decrease to a value of 0.0 before 24:00:00. Only one decreasing time period per schedule will be allowed.

Description: Enter an optional description for this control element.

Exponential Delay

This control element provides the ability to simulate time delays associated with the ramping up/down of system components changing between states, e.g., the spin down of a fan or the opening/closing of a damper. The *exponential delay* element allows you to define an exponential delay based on a time constant.

Node Name: This is an optional name you can provide for this control node. This name can be used to reference this node with a *Phantom control node*.

Time Constants:

> **Increase:** Enter the amount of time it should take for the *output* signal to exponentially increase by (1 – 1/e) % of the total change in the *input* signal when it rises from one state to the next. The format is hh:mm:ss.
>
> **Decrease:** Enter the amount of time it should take for the *output* signal to exponentially decrease by (1 – 1/e) % of the total change in the *input* signal when it falls from one state to the next. The format is hh:mm:ss.
>
> NOTE: An amount of time of about 4.6 times the value entered above is required for the *output* signal to increase/decrease by 99 % of the total change in the *input* signal.

Description: Enter an optional description for this control element.

Maximum and Minimum

The *output* will be the *maximum* or *minimum* of all *input* signals to the control node each time step. Once a node is defined to be of this type, up to three input signals can be drawn directly into it. More signals can be cascaded through the use of *phantom* control elements.

Node Name: This is an optional name you can provide for this control node. This name can be used to reference this node with a *Phantom control node*.

Description: Enter an optional description for this control element.

Integrate over time

The *output* will be the integration over time of *input signal 1* (horizontal) controlled by *input signal 2* (vertical). Integration occurs when *input signal 2* > 0; there is no integration when *input signal 2* = 0; the integral is reinitialized to zero when *input signal 2* < 0. A simple trapezoidal integration method is used.

Running average

The *output* will be the average of the *input* signal integrated over the **Time Span**, Δt_{int}:

$$out = \int in \, dt \, / \, \Delta t_{int}$$

At the start of simulation, when the simulated time is less than the time span, the output will be the average of the input up to that time.

Node Name: This is an optional name you can provide for this control node. This name can be used to reference this node with a *Phantom control node*.

Time Span: Enter the amount of time included in the running average, Δtint. The format is hh:mm:ss.

Description: Enter an optional description for this control element.

Continuous Value File and Discrete Value File (CVF/DVF)

These control elements allow you to implement general schedules by obtaining input values from a file. This allows you to create schedules that are not restricted by the 12-*day schedule* limit of CONTAM *week schedules*. These are text files that you create according to the format specified in the CVF Format and DVF Format sections of this document. You can only use one of each file per simulation, but the file may contain multiple lists of values. Value lists are referenced by *Value name*, which are column headings in the files.

CVF and DVF files differ not only in format but also in behavior. During a simulation, CONTAM will interpolate between the CVF values but not so for DVF values, i.e., they will remain constant until a new value is encountered within the file.

Beginning with CONTAM version 3.2, these files can now be utilized without the use of an associated control node/network, i.e., building components that may be associated with a schedule can do so by directly associating them with a *value name* of a CVF/DVF file. This is now apparent in the Schedule properties of each component within ContamW.

Value Name: Select the name of a set of values from the list of headings as they appear in the CVF/DVF file.

File Name: This field displays the CVF/DVF file that contains the data that will be used by the control node during simulation. You must use the **Data→Controls→Continuous Values File…** and **Data→Controls→Discrete Values File…** menu items to select the files you want to use prior to creating the CVF/DVF control nodes.

Node Name: This is an optional name you can provide for this control node. This name can be used to reference this node with a *Phantom control node*.

Continuous Values File (CVF) Format

You can use CVF files to implement schedules that are not limited to the twelve day types as are CONTAM schedules. CVF values are made available to ContamX through the continuous values file which has a .CVF file name extension. During simulation, ContamX will linearly interpolate between the values provided in the CVF file. You create the text file according to the following format using an external program.

The first line of the CVF is used to identify the type of file. It is exactly:

```
ContinuousValuesFile ContamW 2.1
```

The second line is a description of the file entered by the user:

```
desc[]          // file description (I1) {W}; may be blank
```

The next line defines the period covered by the file:

```
StartDate       // first date for DEF data (mm/dd → IX)
EndDate         // last date for DEF data (mm/dd → IX)
```

The next section specifies the control node names:

```
_nbvfn          // number of CVF node names
```

followed by _nbvfn lines consisting of node names:

```
name[]          // node name (I1)
```

The remainder of the file consists of data for all nodes for each date and time from StartDate to EndDate:

```
date            // date (mm/dd → IX)
time            // time of day (hh:mm:ss → I4)
value[1]        // value [?] for name[1] (R4)
...
value[_nbvf]    // value [?]for name[_nbvf] (R4)
```

The node data must start at time 00:00:00 on the StartDate and end at 24:00:00 on the EndDate. The times must be in consecutive order, but the difference between successive times need not be constant.

The file description may not begin with a "`!`". The `StartDate` and `EndDate` are used to verify that the file data covers the entire period to be simulated. The `StartDate` may not be later than the `EndDate`. Data elements on a single line are separated by tabs. The data must be in time-sequential order. The file values must be in the units needed for the signal created by the control node.

Node names may not include imbedded blanks. Data for nodes that are not in the project file will be ignored. If a CVF node name in the project file is not included in the CVF, a fatal error will result. ContamW will assist the user by checking node names. ContamX will perform the name check before simulation begins.

Discrete Values File (DVF) Format

You can use DVF files to implement schedules that are not limited to the twelve day types as are CONTAM schedules. DVF values are made available to ContamX through the discrete values file which has a DVF file name extension. As the name implies, ContamX will <u>not</u> interpolate between the values provided in the DVF file. You create the text file according to the following format using an external program.

The first line of the DVF is used to identify the type of file. It is exactly:

```
DiscreteValuesFile ContamW 2.1
```

The second line is a description of the file entered by the user:

```
desc[]          // file description (I1) {W}; may be blank
```

The next line defines the period covered by the file:

```
StartDate       // first date for DEF data (mm/dd → IX)
EndDate         // last date for DEF data (mm/dd → IX)
```

The next section specifies the control node names:

```
_ndefn          // number of DVF node names
```

followed by _ndefn lines consisting of:

```
name[]          // node name (I1)
value           // initial value [?] (R4)
```

Succeeding lines present data whenever a node value changes:

```
date            // date (mm/dd → IX)
```

```
time            // time of day (hh:mm:ss → I4)
index           // index into name[]
value           // new value [?] (R4)
```

The file description may not begin with a "**!**". The `StartDate` and `EndDate` are used to verify that the file data covers the entire period to be simulated. The `StartDate` may not be later than the `EndDate`. Data elements on a single line are separated by tabs. The data must be in time-sequential order. More than one node may change at the same time. The file values must be in the units needed for the signal created by the control node.

Node names may not include imbedded blanks. Data for nodes that are not in the project file will be ignored. If a DVF node name in the project file is not included in the DVF, a fatal error will result. ContamW will assist the user by checking node names. ContamX will perform the name check before simulation begins.

3.8.8 Control Super Elements

Prior to CONTAM version 2.2, ContamW required you to draw all control networks directly on the main SketchPad even if they shared the same logic and control elements, leading to the potential for much repetitive work. In order to minimize user input and to allow sharing of control logic between different project files, controls have been enhanced to enable the creation of *Control Super Elements*. *Super Elements* can be saved in the project file as well as in a new CONTAM Library file type LB5.

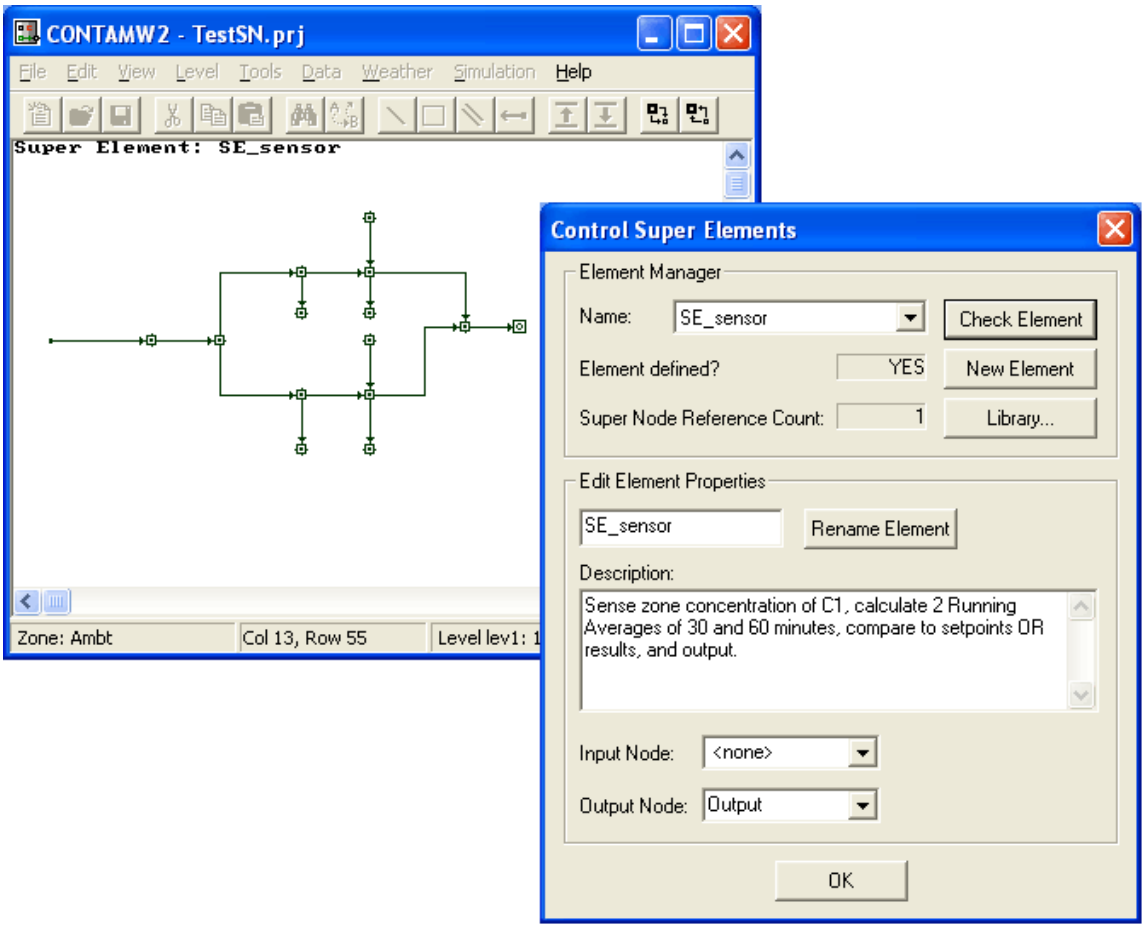
Terminology

Super Element **-** named control logic diagram and accompanying control elements (*sub-nodes*) from the previously presented list of control elements. In object-oriented programming terms this is similar to a class in that it is not actually used until it is instantiated as an object.

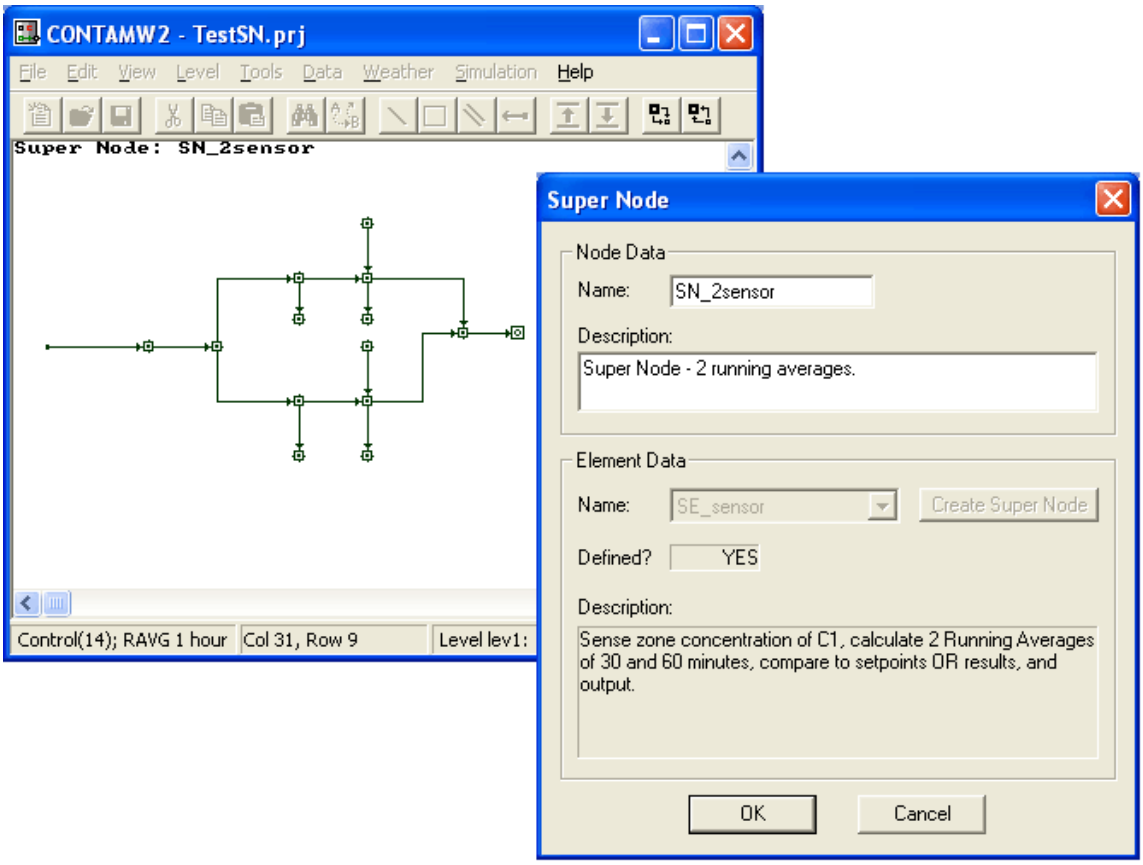
Super Node **-** an instantiated *Super Element*. In object-oriented programming terms this would be an object or instance of a user-defined class.

Control Super Element / Super Node SketchPad

With the advent of the new *control super element*, comes another use of the SketchPad to create and edit *super elements* and *super nodes*. When working with super elements only the controls drawing tool is enabled, along with the ability to define the control network icons. This SketchPad is activated via the **Data→Controls→Super Elements...** menu item. When the Super Element SketchPad is active, the upper left corner of the SketchPad will display "**`Super Element:`**" followed by the name of the super element currently displayed on the SketchPad. When working with super elements the "Control Super Elements" dialog box will also be displayed as shown below.



The Super Node SketchPad enables only the modification of existing control sub-node icons, so the control drawing tool will be disabled as will the ability to delete control network icons. The Super Node SketchPad is activated by *instantiating* an existing *control super element* or double-clicking on a Super Node icon. When the Super Node SketchPad is active, the upper left corner of the SketchPad will display "**`Super Node:`**" followed by the name of the super node currently displayed on the SketchPad. When working with super nodes the "Super Node" dialog box will also be displayed as shown below.



Defining Super Elements

To create a super element you activate the Super Element SketchPad. Unlike other library element types, all super elements must be created and edited within a CONTAM project file. Initiating the super element editing mode activates both the Super Element SketchPad and the Control Super Elements dialog box shown below. Use the SketchPad to draw the *super element diagram* and define control sub-node types in much the same way as is done for controls on the main SketchPad. The dialog box is provided to name, save and set other super element properties described below.

Deleting Super Elements

You can delete super elements via the CONTAMW Library Manager. Only those super elements that are currently not referenced by a Super Node can be deleted. You can determine whether or not a super element is referenced by the *super node reference count* provided on the Control Super Element dialog box when displaying a particular super element on the Super Element SketchPad.

Modifying Super Elements

Modifying Super Elements consists of modifying the drawing (i.e., adding and deleting control links and sub-nodes), modifying the properties of individual sub-nodes and modifying the Super Element Properties.

> **Modifying the drawing** – You can only modify the drawing if the *super node reference count* is 0. This limitation is in effect, because all instances of a *super element* (i.e., *super nodes*) reference the sketch data and list of sub-nodes of the super element.

Modifying sub-node properties – You can modify the properties of sub-nodes as you desire. This will not affect existing instances of the super element; however, this will change the default properties of new instances of the super element.

Modifying Super Element properties – You can modify the super element properties as you so desire. This will not affect existing instances as super nodes simply reference the super element properties, but do not create new instances of them.

Control Super Element Properties

This section provides information on utilizing the Control Super Element dialog box that together with the Super Element SketchPad enables you to manage and define control super elements and is activated via the **Data→Controls→Super Elements...** menu item. There are two section of information presented: Element Manager and Element Properties editor.

Element Manager

This is used to select super elements for editing and creating new elements and activating the CONTAMW Library Manager to work with Super Element Library files.

Name: This combo box provides you with the name of the currently active super element for which the sketch data is also displayed on the Super Element SketchPad. The name should also appear on the upper left corner of the sketchpad. You can also select other existing super elements from the drop down list. This will change the super element displayed upon the sketchpad as well as the properties displayed on this dialog box.

Check Element: While working with a super element, you can perform a check on it to determine if is a fully defined super element. If the check passes then the super element is considered to be fully defined. If the check fails, then ContamW will provide feedback in the form of messages and highlighting of errors on the sketch. Even though a super element is undefined, it can still be saved in its incomplete form.

There are a few basic reasons why the check could fail. Super elements are very similar to control networks on the main SketchPad, so undefined super elements can result from problems similar to main SketchPad control network as well as problems specific to super elements.

Potential reasons for undefined Super Elements:

- Undefined sub-nodes (i.e., undefined sketch icons)
- Incorrect number of inputs to control nodes
- Circular control logic
- No Output Node set
- Pre-defined input to Input Node

New Element: This will create new super element having a default name of "SE_##" where ## will depend on the number of existing super elements. New super elements have no sketch data, so the SketchPad will be cleared. Once you create a super element, you do not have to save it. Any modification you make to the sketch and properties will be maintained automatically with the exception of the name which you must set via the "Rename Element" feature.

Library: Use this feature to activate the CONTAMW Library Manager to create control library files, copy super elements to and from super element library files (type LB5) and delete unused super elements from the current project file. (See Working with Data and Libraries)

Element defined? This feature indicates whether or not a super element is fully defined (as determined by ContamW) meaning it can be instantiated to create super nodes. Element definition is verified by "checking." Checking is performed using the "Check Element" feature described above, when you switch the currently displayed super element from the list of names, or when you close the Control Super Elements dialog box, i.e., exit the super element editing mode.

Super Node Reference Count: This is the number of instantiated super nodes that reference, the super element. The super element can not be deleted from the project file unless this count is equal to zero.

Edit Element Properties

These are the user-defined properties of the super element other than the sketch and control nodes. These properties include the Name, Description, Input Node name and Output Node name.

Rename Element: Use this feature to enter a new name for the super element and change it with the accompanying button. The new name will be compared to existing names before being reset.

Description: Provide a detailed description of the super element of up to 255 characters. This description will be displayed in the CONTAMW Library Manager. The description and the super element name are currently the only means by which to identify the super elements when viewing them in the library manager.

Input Node: The *input node* of the super element is the sub-node which will receive an input signal if one is drawn to an instantiated supernode on the main SketchPad. This is not required to be set. If needed, select a sub-node to serve as the main input for the super element from the list of *named sub-nodes*. Note that you must have provided a name for the sub-node which you wish to make the *input node*, and the sub-node must be of a type that accepts at least one input.

> Multiple inputs
>
> If you require a super element with multiple inputs, you can define a sub-node to be of type *phantom*. Once you instantiate a super node from the super element, you can select a named control element that you want the *phantom sub-node* to reference.

Output Node: The *output node* of the super element is the sub-node from which the signal will be provided to an output drawn from an instantiated *super node* on the main SketchPad. This is required to be set for a super element to be *defined*. Note that when instantiated, the output node will assume the name of the super node for the purposes of referencing with phantom nodes on the main SketchPad.

Control Super Nodes

Super nodes are instances of *super elements* having the same control logic and set of sub-nodes as the super element. When you instantiate a *super node* from a super element, it takes on the default values of the super element's sub-nodes; however, you can modify the values of the sub-node parameters as needed.

Creating Super Nodes

To create a super node, you select the *Super element...* from the Control Element Types dialog box just as you would define any other control node on the main SketchPad (See Defining Controls). This will activate the Super Node SketchPad mode and the accompanying Super Node dialog box. You then work with both the dialog box and sketchpad to select a super element to instantiate and to modify the sub-nodes as you require for the particular instance (See Control Super Node Properties).

Once you have created the *super node* and exited the super node editing mode, the super node will appear as a single icon (*super node icon*) on the main SketchPad.

Deleting Super Nodes

You delete super nodes from the main SketchPad by simply highlighting the super node icon and pressing the delete key on the keyboard or selecting **Edit→Delete** from the main menu. If any of the sub-nodes of the deleted *super node* are referenced by phantom nodes, you will be informed and prompted as to whether you want to undefine the phantom nodes (and sub-nodes) that reference them.

Modifying Super Nodes

Once you have instantiated a *super node*, you can modify the sub-nodes of a *super node*. Activate the super node editing mode by double-clicking the super node icon on the main SketchPad. Simply double-click on the sub-nodes whose properties you wish to modify and edit them as you would nodes on the main SketchPad. Note that you can not modify the sketch in any way, i.e., you can not draw or delete sub-nodes.

Control Super Node Properties

This section provides information on utilizing the Super Node dialog box that together with the Super Node SketchPad enables you to manage and define control super nodes. This dialog box is activated when creating a new *super node* or when modifying an existing one. There are two section of information presented: Node Data and Element Data.

Node Data

This displays *super node* specific data used to identify a specific super node.

Name: As with all control node types, you can provide a name for the *super node*. Named super nodes can be referenced by *phantom nodes*. When you name a super node, its output sub-node will also be given the same name, because the output sub-node will be the node actually referenced by any *phantom nodes* that refer to the *super node*.

Description: Enter a detailed description of the super node to help you identify the super node. This description will appear in the status bar when you highlight the super node icon on the SketchPad.

Element Data

This section enables you to select and set the super element from which to instantiate a new super node. Once instantiated this section simply provides information on the super element from which the super node was instantiated.

Name: Prior to instantiation of a super node, this combo box provides you with the name of the currently active super element for which the sketch data is also displayed on the Super Element SketchPad. The name should also appear in the upper left corner of the SketchPad. You can also select other existing super elements from the drop down list. This will change the super element displayed upon the sketchpad as well as the properties displayed on this dialog box.

Create Super Node: Click this button to instantiate a *super node* from the currently selected *super element*. Once instantiated, this button will be disabled along with the Name drop down combo box.

Note that when creating a new super node, until you *create the super node*, any changes you make to sub-nodes of the displayed super element will modify sub-nodes of the *super element* and not a *super node*.

Defined?: This indicates whether or not the currently selected *super element* is fully defined – if "YES" then a *super node* can be instantiated from the super element, otherwise not.

Description: This displays the detailed description of the currently displayed *super element*.

3.9 WORKING WITH SPECIES AND CONTAMINANTS

Airborne contaminants are dispersed throughout buildings due to multiple transport mechanisms. These include air movements into, out of, and within a building system; heating, ventilating, and air-conditioning (HVAC) systems; removal by filtration, deposition, and sorption; generation and resuspension; chemical reaction; and radio-chemical decay. CONTAM provides you with the ability to simulate these methods of dispersal of airborne contaminants within built structures.

Species is a general term used (by CONTAM) to identify substances that can be used as contaminants during a simulation, i.e., contaminants are those species you designate to be used in a simulation. You can include as many species/contaminants as you need within a project.

CONTAM handles both *trace and non-trace contaminants*. Trace contaminants are those that exist at concentrations that do not cause a "significant" change in the density of air. Non-trace contaminants are those that can affect the density of the air. The mass fractions of non-trace contaminants will always sum to 1 kg/kg, i.e., they are the constituents of the air.

In versions prior to CONTAM 2.4 an underlying assumption of the model was that each zone is considered to have a single concentration value at any given time. This assumption is sometimes referred to as the "well-mixed" zone assumption. With respect to contaminant analysis, this means that there are no spatial concentration gradients within a given zone. However, beginning with version 2.4, zones can be pre-configured by the user to be one-dimensional convection/diffusion zones in which contaminants can vary along a user-defined axis and simulated using the *short time step* method (see Run Control Properties and One-Dimensional Convection-Diffusion Flow**Error! Reference source not found.**). Beginning with CONTAM version 3.0, a single region of a building/project can be considered as a computational fluid dynamics (CFD) zone (Detailed Zone Properties). A CFD calculation is used to determine three-dimensional pressure, airflow and contaminant concentration fields within the CFD zone.

You can use CONTAM to perform steady state, transient, and cyclical contaminant analysis. Steady state analysis refers to the determination of contaminant concentrations under steady flow conditions. Transient analysis refers to the determination of the time history of contaminant concentrations within each zone for a set of discrete time-steps. With a transient analysis you can implement the effects of varying ambient contaminants using a contaminant file (See Contaminant Files). A cyclic analysis determines the time history of contaminant concentrations for a 24 h period such that the concentrations at the beginning and end of the 24 h period are the same for each zone.

With CONTAM you must define contaminants in order to utilize the features that CONTAM provides to simulate the generation of contaminants by sources, the removal by sinks (including filters and kinetic reactions), and to determine occupant exposures. Contaminants must be created/defined prior to implementing any of these contaminant-dependent features of CONTAM. Prior to implementing contaminants within a project, you can first define the building layout on the SketchPad, define all airflow paths (including ducts) and test your building model by performing a steady state airflow simulation.

3.9.1 Non-trace Contaminants and Water Vapor

CONTAM handles non-trace contaminants, which are those contaminants that you define to be the constituents of air. Typically, you might want to treat water vapor as a non-trace contaminant. CONTAM will treat a contaminant with the name "H20" as water vapor. The

concentration will be affected by the humidity ratio contained in weather files that you use during simulations (See Defining Transient Weather and Wind). ContamX will calculate the concentration of H_2O within each zone, as the mass fraction of water to air in units of *kg of* H_2O */kg of air*, where *kg of air = kg of dry air + kg of* H_2O. Whereas the humidity ratio is typically given as the ratio of the mass of water vapor to that of dry air. You can convert between the two using the following relationship.

Humidity ratio = mass fraction of H_2O / (1 – mass fraction of H_2O)

For example, to create air from its typical constituents you might define the following set of non-trace contaminants (See Creating Species and Contaminants):

Name	Molar Mass [kg/kmol]	Default Concentration [kg_cont/kg_air]	Trace Contaminant	Use in Simulation
N2	28	0.7554	Non-trace	Use
O2	32	0.2314	Non-trace	Use
Ar	40	0.0127	Non-trace	Use
CO2	44	0.0005	Non-trace	Use
H2O	18	0.0000	Non-trace	Use

Note in the above example, the ∑Default Concentration = 1.0 kg/kg (see Project Species for details).

As a simpler example, you might create a dry air contaminant, Air, and a water vapor contaminant, H20, as follows.

Name	Molar Mass [kg/kmol]	Default Concentration [kg_cont/kg_air]	Trace Contaminant	Use in Simulation
DryAir	28.9645	1.000	Non-trace	Use
H2O	18.0150	0.000	Non-trace	Use

3.9.2 Creating Species and Contaminants

You can create species by either activating the Project Species dialog box via the **Data → Contaminants…** menu or using the CONTAMW Data and Library Manager: Species dialog box accessed via the **Data → Species…** menu. The Project Species dialog box will enable you to create and modify species for the current project. The Library Manager will enable you to import species from species-related libraries. A detailed description of each of the species/contaminant properties is given in the Species Properties section of this manual. Each species has a *default concentration* associated with it that you define (or leave zero). You can use this default concentration to set the initial concentrations within a building without having to utilize a source element. This default value will also be used as the ambient contaminant concentration unless overridden by values in either a contaminant file or a WPC file (See See Howard-Reed [2004] and See Howard-Reed [2004]).

3.9.3 Deleting Species and Contaminants

You can delete species using the Delete button of the Project Species dialog box or of the CONTAMW Data and Library Manager: Species dialog box. Species can not be deleted if they are associated with a source/sink, filter or kinetic reaction. For those species for which an association exists, the Delete buttons will be deactivated. You can also disable the use of a contaminant within a simulation by deselecting the "Use in Simulation" box for a given species from within the Species Properties dialog box.

3.9.4 Modifying Species and Contaminants

Once you have created species for a project, you can modify any of the related parameters. This is done by using the Edit button from within the Species Properties dialog box that is accessed via the **Data → Contaminants…** menu item. ContamW will allow you to change the name of an existing contaminant without affecting any of the currently associated sources, sinks, filters or kinetic reactions as they reference contaminants by an internal index created by ContamW.

3.9.5 Project Species

Species must be defined before any related elements (e.g. source/sinks, filters and kinetic reactions) can be defined. From the Project Species page any previously defined species may be viewed, edited, or deleted and new species may be defined. The following describes the information that appears on the Project Species dialog box.

Species: This is a list of the species that are currently defined within the current project.

Species Properties: These are the properties of the currently highlighted species.

Contaminants: This is a list of the project species for which the "Use in Simulation" property is set to "Use," i.e., these are selected to be contaminants in the simulation.

Contaminant Summary: This is a summary of contaminant-related information including the number of non-trace contaminants and the summation of the default concentrations of the non-trace contaminants which must be 1.0.

NOTE: *If the project contains water vapor, H2O, then its default concentration is not included in this summation at this point, however, it will be accounted for by ContamX during simulation if initial concentrations are set to ambient (see Simulation - Airflow Numerics Properties).*

New, Edit and Delete: Click the "New" button to create a new species, or use the "Library..." button to import/export species from/to a species library file. To modify the properties of an existing species, highlight the species in the species list and click the "Edit" button. Click the "Delete" button to delete the selected species.

IMPORTANT: Changing the number of contaminants, i.e. those species used in the simulation, will reset the initial concentrations of all zones to the default contaminant concentrations. You can reset all of these concentrations via the Run Control Properties of the simulation parameters or via the Contaminant Data properties of the individual zones.

3.9.6 Species Properties

This section provides detailed descriptions of the specific species properties.

Name: You will use this name to refer to the species throughout the program. You must give each species a name that is unique to the current project.

Molar Mass: The molar mass in g/mol or kg/kmol. CONTAM uses this value to convert volumetric concentrations to mass fractions and to calculate the gas constant of the air mixture made up by non-trace contaminants. You must enter a value of at least 1.0.

Default Concentration: CONTAM will apply this value as the default initial concentration for each zone that you create. You can revise the initial concentration for individual zones as per your requirements. This is also the ambient contaminant concentration during simulations which do not utilize contaminant data from one of the external contaminant data files (CTM or WPC).

Diffusion Coefficient: This value will be used during simulations using the *1D Duct Model* of the *Short Time Step Method* (See Run Control Properties in the Working with Simulations section). The dispersal coefficient will be calculated from this value depending on the Reynolds Number in a given duct segment.

Mean Diameter: Enter the mean particle diameter if this species is to be a particulate type species. CONTAM utilizes this value when converting between particle count units and particle mass and volume units and to determine filter efficiency of Simple Particle Filter elements for this size particle (See Simple Particle Filter in the Working with Filters section).

NOTE: This is not necessarily meant to be, for example, the aerodynamic diameter as the current contaminant source models do not treat particles different from gaseous contaminants.

Effective Density: Enter a density that you want CONTAM to use as the effective density of a species you want to consider to be a particulate type species. Currently, CONTAM only utilizes this value when converting between particle count units and particle mass and volume units.

Specific Heat: The specific heat of a contaminant will used during simulation only when the *Variable Junction Temperatures* method is selected.

Decay Rate: This is an exponential decay constant based upon the half-life of the radioactive species calculated using the following equation:

Decay Rate = $\ln(2) / \tau_{1/2}$,

where $\tau_{1/2}$ is the half-life of the radioactive species in seconds.

ContamW will use this value to perform unit conversions for radioactive contaminants. Radioactive species are only distinguished from other species by the units associated with them, not by modeling the radioactive decay process. To model a radioactive decay, use a kinetic reaction (See Kinetic Reaction Data).

UVGI Susceptibility Constant: A species-dependent rate constant used to determine UVGI filter effectiveness for a specific contaminant [m^2/J]. Values for several species are provided in [Kowalski, et al. 2000] and they range from 10^{-3} $cm^2/\mu J$ to 10^{-6} $cm^2/\mu J$. (new for version 3.2)

Non-Trace Contaminant: Non-trace contaminants are those that can affect the density of the air, i.e., are considered components of the air. (See Non-trace Contaminants and Water Vapor)

Use in Simulation: This determines whether or not a species will be a contaminant. Checking "Use in Simulation" indicates that a species will be a contaminant and used in the simulation.

Description: Use this to provide a more detailed description of the species.

Guideline Value: Currently not implemented.

Guideline Description: Currently not implemented.

3.9.7 Contaminant Files

You can account for changes in the ambient contaminant concentrations when performing transient contaminant simulations using ambient contaminant (CTM) files. If you associate a contaminant file with your project (see Run Control Properties) ContamX will compare those contaminants defined within your project file with those contained in the contaminant file. If they match, ContamX will use the ambient concentrations in the contaminant file within the simulation, otherwise they will be ignored.

You create the contaminant files external to ContamW. You can use a utility program provided by NIST, Weather 2.0, to convert existing CONTAM 1.0 weather (which previously contained ambient contaminant concentrations) files to CONTAM 2.0 weather and contaminant files.

Note that with this method the ambient concentration does not vary spacially, only temporally. If you require spacial variations of outdoor contaminant level, then refer to the section entitled Working with WPC Files.

Contaminant (CTM) File Format

The CONTAM contaminant (species) file is a tab-delimited ASCII text file. These files are easily created, imported and modified using common spreadsheet applications. They can be saved as tab-delimited text files for use with CONTAM. In the listing below `I1` indicates a character string, `I2` indicates a short integer and `R4` indicates a four-byte real number. Individual lines of data are separated by dashes (these dashes are only shown here for format presentation purposes and should not be included in the actual files). Each day must begin with time 00:00:00 and end with time 24:00:00. The time interval between 00:00:00 and 24:00:00 can be either regular or irregular – ContamX will interpolate as necessary. Comments are allowed and are indicated by an "!" (exclamation point). Anything that appears on a line following a comment indicator is ignored by the contaminant file reader.

```
SpeciesFile ContamW 2.0 ! file and version identification
-----
I1 Descr[] ! species file description
-----
I1 StartDate ! day of year (1/1 - 12/31)
I1 EndDate ! day of year (1/1 - 12/31)
I2 NumCont ! number of species
-----
I1 Name[1] ! name of species 1 (16 characters max)
I1 Name[2] ! name of species 2
```

```
...
I2 Name[NumCont] ! name of species NumCont
-----
!Date Time Name[1] Name[2] ... Name[NumCont]
-----
! for each day: start_date – end_date
! for each time: 00:00:00 - 24:00:00 increment may be variable
 I1 Date ! date (1/1 – 12/31)
 I1 Time ! time of day 00:00:00 – 24:00:00
 R4 Conc[1] ! concentration of species 1
 R4 Conc[2] ! concentration of species 2
 ...
 R4 Conc[NumCont] ! concentration of species nc
 ! [kg_species/kg_air]
-----
```

Editing Contaminant Files

You can import ambient files into a spreadsheet program for editing. However you must make sure that the date format is compatible with CONTAM when you save the file. One well-known spreadsheet program will convert the CONTAM date format of mm/dd into the mmm-dd form. For example the CONTAM date format for January 1 is "1/1", but the spreadsheet program will convert it to "Jan-01" unless you "force" it to import the first column of data – the one that contains date information – as text. Another option is to accept the default importing option then change the cell format for each date to the CONTAM format mm/dd prior to saving the file as tab-delimitted.

Contaminant Unit Conversions

Gaseous contaminants – to convert from ppm to:

$kg/kg = (ppm \times MW) / (1\,000\,000 \times V_s \times \rho_{air})$

$mg/L = (ppm \times MW) / (1\,000 \times V_s)$

$mg/m^3 = (ppm \times MW) / V_s$

$mg/ft^3 = (ppm \times MW) / (35.3147 \times V_s)$

Particle Contaminants – to convert from $\#/m^3$ to kg/kg:

$$\frac{kg_{cont}}{kg_{air}} = \frac{\#}{m^3} \times \frac{\rho_{eff}}{\rho_{air}} \frac{1}{6} \pi D_{mean}^3$$

where:

MW = molar mass of contaminant [kg/kmol]

V_s = 24.05 m^3 which is the volume of one kmol of air at T = 20 ºC and P = 101 325 Pa (1 atm)

ρ_{air} = 1.204 kg/m^3 at T = 20 ºC

ρ_{eff} = effective particle density [kg/m^3]

D_{mean} = mean particle diameter [m]

3.10 WORKING WITH SOURCES AND SINKS

Sources and sinks are used to generate contaminants within and remove contaminants from zones. Sources can be used to simulate building materials such as carpet, pieces of furniture, and episodic events such as the use of cleaning products. You can define an almost unlimited number of sources within a project.

To implement a source/sink, you must first define the species with which you want the source/sink to be associated. You then place a source/sink icon within a zone on the SketchPad. Once you place source/sinks on the SketchPad, you can define their characteristics, move, copy, and delete them. You must associate each source/sink with a predefined species and provide specific information that describes its emission and/or removal characteristics.

NOTE: It is important to note that even though you associate a species with each source/sink, only those species that are designated to be contaminants will actually be used when performing simulations (See Species Properties).

Much of the information that describes the characteristics of a source/sink is contained in *source/sink elements* that you must create when defining source/sinks. The details of drawing, defining and modifying sources and sinks are described in the following sections.

3.10.1 Source/Sink Elements

CONTAM uses mathematical relationships referred to as *source/sink elements* or models to implement sources and sinks when performing contaminant simulations. CONTAM can implement several source/sink models to generate contaminants within or remove contaminants from a zone. Depending on the source/sink model, it could be used to emulate either a source, a sink or both. Each of these source/sink element types is described in detail in the Source/Sink Element Types section. Every source/sink must refer to a single *source/sink element*; however, multiple source/sinks can refer to the same *source/sink element*.

Source/sink elements can also be stored within a CONTAM library file and shared between different CONTAM project files (See Working with Data and Libraries).

3.10.2 Creating Sources and Sinks

You create source/sinks by placing a source/sink icon on the SketchPad (See Drawing Building Component Icons). You must first define at least one species before ContamW will allow you to place a source/sink icon upon the SketchPad. Each source/sink must be placed within a building zone. You cannot place them in the ambient zone. Ambient sources are handled using either the initial concentration property of a contaminant (for steady state simulations) or the concentration contained within a weather file (for transient simulations). The "Source/sink" menu selection will be disabled (grayed out) if you pop-up on a location where ContamW does not allow the icon to be placed.

3.10.3 Deleting Sources and Sinks

You delete source/sink icons using the icon deletion procedure (See Deleting Building Component Icons). Deleting source/sink icons from the SketchPad does not delete any source/sink elements that you may have defined. These elements can only be deleted using the CONTAMW Library Manager, accessible through the **Data** → **Source/Sinks...** menu.

3.10.4 Modifying Sources and Sinks

To modify the parameters of a source/sink, you use the icon definition procedure (See Defining Building Component Icons) to display its properties and make the desired changes. You can also move and copy source/sinks.

Moving Source/Sink Icons

Once you define a source/sink, you can move the icon using the **Cut** and **Paste** functions of the **Edit** menu. You can move the source/sink icon anywhere within the zone in which it is currently located.

Copying Source/Sink Icons

Once you define a source/sink, you can copy the icon using the **Copy** and **Paste** functions of the **Edit** menu. You can copy a source/sink to any blank cell within a zone on any level of a project.

3.10.5 Defining Sources and Sinks

After you draw a *Source/Sink icon* on the SketchPad you must define it using the icon definition procedure (See Defining Building Component Icons) to display and edit the properties. This will display the "Source/Sink Properties" property sheet. Detailed descriptions of all source/sink properties are given in the Source and Sink Properties section of this manual. Once you have defined the properties, the icon will be displayed in black. From now on, you use this icon to access the properties of the source/sink.

Among the properties associated with a source/sink are optional *multiplier* and *schedule* information. You would use the multiplier to proportionally increase the generation or removal of a contaminant for a particular source/sink. For example, if you have created a *source/sink element* with a certain emission rate per square meter of surface area (such as a floor covering), you could use this multiplier to account for the surface area of the source/sink for each zone in which you place a source/sink implementing this *source/sink element*. You could use a *schedule* to vary the emission/removal characteristics of a source/sink with time. For example, you could provide a *schedule* for an episodic source (burst source) that would periodically cause the release of a contaminant when performing a transient simulation.

3.10.6 Source and Sink Properties

This section provides detailed descriptions of the specific source/sink properties. Each of the following subsections are the context-sensitive help topics that you can access by pressing **F1** when working with property pages of the "Source/Sink Properties" property sheet.

Source/Sink - Source/Sink Element Properties

Source/Sink Number: This is the number that appears in the status bar when you highlight an icon. ContamW automatically assigns this number to each *source/sink* once they are defined. This number could change as you add and remove source/sinks to or from a project. Each time you save a project, the source/sink numbers are reassigned. Numbers are assigned beginning on the top level, starting from the upper-left corner of the SketchPad, moving left-to-right and top-to-bottom of the SketchPad.

Source/Sink Element: The source/sink element provides the mathematical model to be associated with the selected source/sink.

Description: Displays the description property of the element selected in the "Name" field.

Name: <required> Select a source/sink element from those contained in the Name list. You can add elements to the list by creating new elements within the project or importing them from a library file. Use the "New Element" button to define a new element. Activate the CONTAMW Library Manager to import elements from library files via the "Library" button (See Working with Data and Libraries). Use the "Edit Element" button to modify the properties of the element displayed in the Name field.

Model Summary: This summarizes the information associated with the source/sink element whose name is currently displayed in the Name field.

Source/Sink - Multiplier and Location Properties

Source/Sink Number: This is the number that appears in the status bar when you highlight an icon.
This is the number that appears in the status bar when you highlight an icon.

Source/Sink Number: This is the number that appears in the status bar when you highlight an icon.

Multiplier: A constant value by which the source strength will be multiplied during simulation. With this feature you could define a *source/sink element* having a source strength per unit area then use the multiplier as the area of the zone for each source/sink that uses the per unit area source/sink element.

NOTE: The multiplier serves as the surface area for the Boundary Layer Diffusion Model, and should be entered in units of m^2.

Initial Concentration: Some source/sinks are reversible in that they enable storage within or on the surface and re-emission or resuspension of contaminant from the surface. These source/sinks actually contribute a contaminant node to the contaminant transport equations. Currently, there are two such element types: *Boundary Layer Diffusion Controlled* and *Deposition with Resuspension*. The *Initial Concentration* properties will be enabled for these element types, and you can set the values and units of these initial loadings.

Location: These are the X, Y and Relative Elevation (Z) coordinates and units for the selected source/sink. X and Y are absolute coordinates and Rel Elevation is relative to the level on which the source/sink is located. To create a point source, set the Minimum and Maximum values to be the same. To create a source which distributes emissions along the 1D axis of the zone, enter Minimum and Maximum values that span the desired region of the zone in which the source is located.

These values are required for any source/sink that is located within a *1D convection/diffusion zone* (see 1D Zone Data). These coordinates will be verified by ContamW against the axis of the 1D zone in which it is located. This verification will be performed automatically when you select either the **Run Simulation** or **Create a ContamX Input File** command from the **Simulation** menu.

CFD Source ID: Each source/sink that is located within a CFD zone must be given a unique identifier. This id will be used to identify the source/sink within the CFD0 Editor that is used to define the location of the interface paths with respect to the CFD zone.

Source/Sink - Schedule Properties

Source/Sink Number: This is the number that appears in the status bar when you highlight an icon.

Schedule: You can use a schedule to modify the source strength as a function of time. Two source/sink element types, *Burst Source* and *Decaying Source*, require the use of a schedule. If there are previously defined schedules within the current project, you may select one from the Name list. Click the "New Schedule" button to define a new schedule. To view or modify an existing schedule click the "Edit Schedule" button. Setting this field to <none> indicates no schedule (See See Howard-Reed [2004]).

CVF/DVF Control: Values from either a CVF or DVF file can be used in place of the above Schedule (read about CVF and DVF controls in the Control Element Types section). This enables using these control element types without having to draw/define an associated control network. The associated radio buttons "Use Neither", "Use CVF" and "Use DVF" are used to select the desired control method. Select the heading name from the associated "Name" drop down box that will display the headings of either the CVF or DVF file as per the aforementioned radio button selection.

Control Status: This will display the current method of control: Schedule, direct CVF, direct DVF or Control Network. If an actuator of a control network on the SketchPad is associated with the scheduled item then it will take precedence over the Schedule and CVF/DVF Control. If either the "Use CVF" or "Use DVF" radio button is selected then CVF or DVF will take precedence over an associated schedule.

Source/Sink Element Types

These are the source/sink element types or models that you can implement with CONTAM. Refer to the related information for each specific model to obtain detailed descriptions of them. For the so-called surface elements, a surface results file can be generated that will provide the transient loadings for each of these surface types (see Results Files in Working with Results).

CONTAM Elements

Constant Coefficient Model: The general source/sink model allows constant contaminant generation and deposition rates.

Pressure Driven Model: This model is for contaminant sources that are governed by the pressure differences between zones.

Cutoff Concentration Model: This model will reduce emissions as the concentration within the zone in which it is implemented approaches a specified cutoff concentration. This model may be appropriate for some sources of Volatile Organic Compound (VOC).

Decaying Source Model: This model provides a source that will exponentially decay with time according to a user-defined time constant. This model may also be appropriate for some VOC sources.

Boundary Layer Diffusion Model: Used to model reversible diffusion of a contaminant through the air-phase boundary layer of a material surface and the sorption of the contaminant into a sink with a linear sorption isotherm.

Burst Source Model: Used to model an instantaneous release of a mass of contaminant within a zone.

Deposition Velocity Sink Model: Used to model contaminant deposition onto a surface as specified by a *deposition velocity*.

Deposition Rate Sink Model: Used to model contaminant deposition onto a surface as specified by a *deposition rate*.

Deposition with Resuspension Model: Used to model contaminant deposition and resuspension to and from a surface.

Super Source/Sink Element: The super element allows multiple elements of the other types to be combined into a single element. This can reduce repetitious work required to create multiple sources throughout a building by allowing multiple sources to be represented by a single icon on the SketchPad.

NRCC Elements

These sources are provided for compatibility with the Material Emissions Database developed by the National Research Council Canada (NRCC) that is largely based on material emission test chamber data [Howard-Reed 2004 and 2006].

Power Law Model: Constant emission rate until specified time after which time the emission rate decays according to the power law relationship.

Peak Model: Constant emission rate until specified time at which the peak emission rate is obtained after which time the emission rate decays according to the peak model relationship.

Source/Sink Element: Constant Coefficient Model

This is a generic source/sink element that provides for direct input of a contaminant generation and removal rate. Units are provided in default calculation units.

$S_\alpha(t)$ = mult·ctrl·G - mult·ctrl·R $C_\alpha(t)$

G = Generation rate [kg_α/s]

R = Effective removal rate [kg_{air}/s]

Determine by multiplying the first-order removal rate of the contaminant [1/s] by air density [kg_{air}/m^3] and volume [m^3] of the zone in which the source is to be located.

$C_\alpha(t)$ = Concentration of contaminant α in the zone [kg_α/kg_{air}]

mult = Source/sink multiplier [no units]

ctrl = Schedule or control value [no units]

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Generation Rate: The rate at which the contaminant is introduced into the zone.

Removal Rate: The rate at which the contaminant is removed from the zone.

Species: Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

Source/Sink Element: Pressure Driven Model

The pressure source/sink model is intended to model contaminant sources that are controlled by the pressure differences between interior and exterior zones, e.g., radon or soil gas entry. Units are provided in the default calculation units.

$S_\alpha(t) = \text{mult}\cdot\text{ctrl}\cdot G\,(\Delta P)^n$

S = contaminant source strength at time t
G = generation rate coefficient [$kg_\alpha/Pa^n\cdot s$]
ΔP = pressure difference (ambient pressure - zone pressure) [Pa]
n = pressure exponent [no units]

mult = Source/sink multiplier [no units]

ctrl = Schedule or control value [no units]

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Generation Rate: This coefficient describes the rate at which the contaminant is introduced into the zone as a function of pressure difference.

Pressure Exponent: Used to describe the dependence on pressure of the contaminant entry.

Species: Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

Source/Sink Element: Cutoff Concentration Model

For volatile organic compounds the source model is sometimes expressed using the cutoff concentration model.

$S_\alpha(t) = \text{mult}\cdot\text{ctrl}\cdot G\,(1 - C_\alpha(t)/C_{cut})$

$S_\alpha(t)$ = Source strength at time t [kg_α/s]

G = Generation rate [kg_α/s]

$C_\alpha(t)$ = Concentration of contaminant α in the zone at time t [kg_α/kg_{air}]

C_{cut} = cutoff concentration at which emission ceases

mult = Source/sink multiplier [no units]

ctrl = Schedule or control value [no units]

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Generation Rate: This coefficient describes the rate at which the contaminant is introduced into the zone as a function of concentration.

Cutoff Concentration: The concentration level where the source ceases its emission.

Species : Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

Source/Sink Element: Decaying Source Model

The decaying source model can be started and stopped multiple times during a simulation using a schedule or control signal. Setting the schedule or control value to zero will turn off the source and a schedule or control value greater than zero will turn the source back on. If no schedule is associated with this source sink, it will activate immediately.

$S_\alpha(t) = \text{mult}\cdot\text{ctrl}\cdot G_0\, e^{-\Delta t/\tau_c}$

where:

$S_\alpha(t)$ = Source strength of contaminant α at time t [kg_α/s]

G_0 = initial generation rate [kg_α/s]
Δt = elapsed time since the start of emission [s]
τ_c = time constant [s]

mult = Source/sink multiplier [no units]

ctrl = Schedule or control value [no units]

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Initial Generation Rate: Contaminant generation is controlled by a schedule. Contaminant generation begins when the schedule changes from a zero to a non-zero value (between zero and one). The initial generation rate is equal to the schedule value times the initial generation rate. A single schedule may be used to initiate several emissions at different times.

Decay Constant: 1/τc, the time at which the generation rate decays by 0.368 of the original rate.

Species: Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

Source/Sink Element: Boundary Layer Diffusion Model

The boundary layer diffusion controlled reversible sink/source model with a linear sorption isotherm follows the descriptions presented in [Axley 1991]. The boundary layer refers to the region above the surface of a material through which a concentration gradient exists between the near-surface concentration and the air-phase concentration. The rate at which a contaminant is transferred onto a surface (sink) is defined by the following equation. Units are provided in the default calculation units.

$S_\alpha(t) = \text{ctrl}\cdot h\cdot d\cdot A_s[C_\alpha(\text{t}) - C_s(t)/k]$

where:

h = film mass transfer coefficient over the sink [m/s]
d = film density of air [kg_{air}/m^3]
A_s = surface area of the adsorbent [m^2]
C_α = concentration in air [kg_α/kg_{air}]
C_s = concentration in the adsorbent
k = Henry adsorption constant or the partition coefficient [$kg_{air}/kg_{surface}$]

ctrl = Schedule or control value [no units]

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Film Transfer Coefficient: The average mass transfer coefficient of the contaminant within the boundary layer (or film) above the surface of the adsorbent.

Film Density of Air: This is the average of the bulk air density and the near-surface density.

Surface Mass: Enter the mass for a unit surface area and use the source/sink multiplier to enter the total area of the adsorbent.

Partition Coefficient: Coefficient that relates the concentration of the contaminant in the bulk-air to that at the surface of the adsorption material.

Species: Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

NOTE: The surface area, A_s, is provided by the source/sink multiplier in units of m^2.

Source/Sink Element: Burst Source Model

The burst model simulates an instantaneous addition of contaminant mass to a zone.

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Mass added to zone: Enter a value for the instantaneous mass release into the zone. The addition of mass is controlled by a schedule; when the schedule changes from zero to a non-zero value, mass is added to the zone in one simulation time step. The amount of mass added is equal to the mass added to zone value times the schedule value. A single schedule can initiate several events at different times.

Species: Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

NOTE: You must associate a schedule for this type of source.

Deposition Velocity Sink Model

This sink model is provided as a convenient means to create a sink using deposition velocities. Units are provided in default calculation units.

$R_\alpha(t) = \text{mult}\cdot\text{ctrl}\cdot v_d\, A_s\, \rho_{air}(t)\, C_\alpha(t)$

where:

$R_a(t)$ = Removal rate (by deposition) at time t [kg_α/s]

v_d = Deposition velocity [m/s]

A_s = Deposition surface area [m^2]

$\rho_{air}(t)$ = Density of air in the source zone at time t [kg_{air}/m^3]

$C_\alpha(t)$ = Concentration of contaminant α at time t [kg_α/kg_{air}]

mult = Source/sink multiplier [no units]

ctrl = Schedule or control value [no units]

This model differs from the Deposition Rate Sink Model in that it is not dependent on the mass of air in the zone in which it is located.

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Deposition Velocity: Enter deposition velocity and units. The air density will be obtained at the time of simulation from the zone in which the source/sink is located.

Surface Area: Enter desired surface area and units. You can enter a value of 1.0 and use the source/sink multiplier to specify the total deposition surface area.

Species: Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

Deposition Rate Sink Model

This sink model is provided as a convenient means to create a sink using deposition rates. This model is dependent on the mass of air in the zone in which the source/sink is located and will behave similar to that of a kinetic reaction having the same reaction rate. Units are provided in default calculation units.

$R_\alpha(t) = \text{mult}\cdot\text{ctrl}\cdot k_d\, V_z\, \rho_{air}(t)\, C_\alpha(t)$

where:

$R_\alpha(t)$ = Removal rate (by deposition) at time t [kg_α/s]

V_z = Zone volume [m^3]

k_d = Deposition rate [1/s]

$\rho_{air}(t)$ = Density of air in the source zone at time t [kg_{air}/m^3]

$C_\alpha(t)$ = Concentration of contaminant α at time t [kg_α/kg_{air}]

mult = Source/sink multiplier [no units]

ctrl = Schedule or control value [no units]

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Deposition Rate: Enter deposition rate and units. The volume and density will be obtained at the time of simulation from the zone in which the source/sink icon is located.

Species: Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

Deposition with Resuspension Model

This source/sink model provides contaminant removal from the zone air by deposition to a surface with source emission via resuspension from the surface. The total deposition surface area remains constant as determined by the (mult·A_s) term in the equation below, but the resuspension

rate and area can be modified by a schedule or control value. Units are provided in default calculation units.

$R_\alpha(t) = v_d\ (\text{mult}\cdot A_s)\ \rho_{air}(t)\ C_\alpha(t)$

$S_\alpha(t) = (\text{ctrl}\cdot r\ A_r)\ L_\alpha(t)$

where:

$R_\alpha(t)$ = Removal rate (by deposition) of contaminant α from zone air to surface at time t [kg_α/s]

v_d = Deposition velocity [m/s]

A_s = Deposition surface area [m^2]

$\rho_{air}(t)$ = Density of air in the source zone at time t [kg_{air}/m^3]

$C_\alpha(t)$ = Concentration of contaminant α in zone air volume at time t [kg_α/kg_{air}]

$S_\alpha(t)$ = Source (resuspension) rate of contaminant α from surface to zone air at time t [kg_α/s]

r = Resuspension rate [1/s]

A_r = Resuspension surface area [m^2]

$L_\alpha(t)$ = Concentration of contaminant α on the deposition surface at time t [kg_α/m^2]

mult = Source/sink multiplier [no units]

ctrl = Schedule or control value [no units]

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Deposition Velocity: Enter deposition velocity and units. The air density will be obtained at the time of simulation from the zone in which the source/sink is located.

Deposition Surface Area: Enter desired surface area and units. You can enter a value of 1.0 and use the source/sink multiplier to specify the total deposition surface area for each specific instance of this element.

Resuspension Rate: Enter the resuspension rate and units.

Resuspension Surface Area: The resuspension surface area may be different from that of the deposition surface area, e.g., a shoe impacting the deposition surface. Once associated with a source/sink, the resuspension rate can be varied using a schedule or control also associated with the source/sink.

Species: Select a previously defined species from the drop down box.

Description: Field for entering a more detailed description of the specific sink/source element.

Super Source/Sink Element

Combine multiple sub-elements into a single element that can be represented by a single icon on the SketchPad.

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Description: Field for entering a more detailed description of the specific sink/source element.

Available Elements: This is a list of basic source/sink elements from which you can build the super element. Items below this list provide information on the currently highlighted element. Use the Edit and New buttons to modify or create basic types of source/sink elements for the list of available elements.

Sub-Elements: Use the Add and Remove buttons to modify this list of sub-elements that make up the super element.

NRCC – Power Law Model

This is an empirically based model provided for compatibility with NRCC Material Emission Database [Howard-Reed 2004 and 2006].

For $t \leq t_p$ the initial emission factor is

$$S_o(t) = \text{mult} \cdot a\ t_p^{-b}$$

and for $t > t_p$

$$S(t) = \text{mult} \cdot a\ t^{-b}$$

NOTE: This type of source can only be triggered once per simulation. If using a schedule, the first non-zero schedule value will trigger the source, otherwise it will begin emitting immediately.

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Initial Emission Factor: Enter the initial emission factor S_0 and its units [kg/s].

Coefficient: This corresponds to a in the above equation. It will be calculated based on S_0, t_p and b. The value of a will be provided by the NRCC Data Base and stored by CONTAM, but the Initial Emission Factor is provided as a more intuitive means for defining the model.

Exponent: Enter a value for the exponent b in the above equation.

PL Model Time: Enter the time t_p until which the initial emission rate applies.

Description: Field for entering a more detailed description of the specific sink/source element.

NRCC - Peak Model

This is an empirically based model provided for compatibility with NRCC Material Emission Database [Howard-Reed 2004 and 2006].

$$S_\alpha(t) = \text{mult} \cdot a \cdot \exp\{-0.5[(1/b) \cdot \ln(t/t_p)]^2\}$$

where:

a, b and t_p are empirical coefficients typically determined from emission chamber measurements.
The emission rate, $S_\alpha(t)$ [kg_α/s], will increase up until time t_p at which point it will begin to decay.

mult = Source/sink multiplier [no units]

NOTE: This type of source can only be triggered once per simulation. If using a schedule, the first non-zero schedule value will trigger the source, otherwise it will begin emitting immediately.

Name: Enter a unique name you want to use to identify the source/sink element. The element will be saved within the current project and can be associated with multiple source/sinks.

Peak Emission Factor: Enter the peak emission factor a. This is the value of $S(t)$ when time $t = t_p$.

Fitting Parameter: Enter a value for the parameter b in the above equation.

Time of Peak: Enter the time t_p at which the peak emission factor is achieved.

Description: Field for entering a more detailed description of the specific source/sink element.

3.11 WORKING WITH FILTERS

CONTAM provides some building components with the ability to incorporate filters that act as contaminant removal devices. Filters consist of two parts: the *filter* itself which is associated with a specific building component (path or duct segment) and *filter elements* that you can associate with one or more filters. Filter elements provide the physical characteristics of the filter and the mathematical relationship that describes the behavior of the filter during a simulation. This is similar in nature to the relationship between airflow paths and airflow elements and between duct segments and duct flow elements. In another sense, the filter is a specific instance of a filter element.

The following building components can incorporate filters and are described as *filter-ready*:

1. Airflow paths
2. Implicit outdoor air and recirculation air paths of simple air-handling systems
3. Supplies and returns of simple air-handling systems
4. Duct segments
5. Duct terminals

3.11.1 Filter Elements

CONTAM uses mathematical relationships referred to as *filter elements* or models to implement filters when performing contaminant simulations. CONTAM can implement several different filter models to remove contaminants as they are transported through building components that have filtering capabilities. These element types are described in detail in the Filter Element Types section. Every *filter* must refer to a single *filter element*; however, multiple *filters* can refer to the same *filter element*.

Filter elements can also be stored within a CONTAM library file and shared between different CONTAM project files (See Working with Data and Libraries).

3.11.2 Creating Filters

You create filters for each building component for which you require a filter. When you are defining one of the *filter-ready* building components presented above, a filter property page will be provided that will enable you to establish whether or not to associate a filter, or to modify or delete a previously defined filter for that component.

3.11.3 Deleting Filters

You can delete filters by accessing the properties of the individual component whose filter you want to delete or by deleting the building component itself. Note that deleting a *filter* does not delete the associated *filter element* (you can not delete a *filter element* that is currently associated with a *filter*).

3.11.4 Modifying Filters

You modify a filter by accessing the Filter property page of the individual building component icon and selecting the "Edit Filter" button to display the "Filter" dialog box.

3.11.5 Filter Properties

Filter Number: This number is automatically assigned by ContamW and can change as you add and remove filters to or from a project. Each time you save a project, the filter numbers are reassigned.

Even though this is the actual filter number used by CONTAM, a different number will be used when presenting filter accumulation results in the *contaminant summary file* (.csm file). This file will present filters according to the building component with which they are associated. For example, filters associated with airflow paths and air handling system supply and return points will be referenced by the path number and the letter "p" to indicate it is a path filter. Filters associated with ducts will be referenced by the duct segment number and the letter "d" or the terminal number and the letter "t" for terminal.

Filter Element

Name: <required> You may select a previously defined filter element from those contained in the Name list. Click the "New Element" button to define a new filter element. To view or modify filter element data click the "Edit Element" button. You can also access the CONTAM Library Manager to import filter elements from library files by clicking the "Library" button (See Working with Data and Libraries). Every filter must have a filter element associated with it.

Description: Displays the description of the filter element currently displayed in the Name field.

Type: Displays the type of filter element for the filter currently displayed in the Name field.

Initial Filter Loading

Total Relative Load: Set the initial loading of the filter relative to the mass of the filter. This can be set to zero to represent a "clean" filter at the beginning of a simulation; otherwise set the value to establish the total mass of contaminants (as a percent of the mass of the filter) accumulated on the filter at the start of a simulation. Relative load is only used for the Gaseous Filter element which is currently the only element for which efficiency is described as a function of filter loading.

3.11.6 Filter Element Types

These are the filter element types or models that you can implement with CONTAM. Refer to the related information for each specific model for detailed descriptions.

Constant Efficiency Filter: Define constant efficiencies for each user-selected species.

Simple Particle Filter: Define filter efficiency as a function of particle size, for example a MERV curve.

Simple Gaseous Filter: Define filter efficiency as a function of filter loading for a user selectable set of species.

Penn State UVGI Filter: Define an ultra-violet germicidal irradiation filter.

Super Filter: This enables the combination of multiple filter elements into a single filter, e.g., a packaged filter that includes a gaseous filter and a pre-filter that filters particles.

Constant Efficiency Filter

The *constant efficiency filter* element is made up of a set of species and associated efficiencies. Each filter element can be associated with anywhere between one and the number of species that exist within the current project. Only those species that you have selected to be contaminants will be accounted for when you perform simulations.

Name: A unique name you want to use to identify the filter element.

Area: The face area and units of the filter element.

Depth: The depth of the filter element along the axis of airflow.

Density: The density of the filter media.

Area, Depth and Density are currently used to calculate the mass of the filter media which is only relevant for the Simple Gaseous Filter model.

Description: Use this field to provide a more detailed description of the filter element.

Edit Element Data: Select to define the specific properties of the filter element.

> **Filtered Species:** This is a list of the species and filter efficiencies for the filter element.
>
> **Edit Filter Data:** Use the *Species* and *Efficiency* fields to set and modify the *Filtered Species* list along with the *Add*, *Replace* and *Delete* command buttons.
>
> **Efficiency:** Use this to enter the *filter efficiency* for the currently highlighted species. Filter efficiencies must be between 0.0 and 1.0.
>
> **Species:** This is a list of all the *species* in either the current project or library file depending on wether you are editing project data or library data. Select the *species* for which you want to set a filter efficiency.

Simple Particle Filter

The *simple particle filter* element consists of a single efficiency vs. particle size curve. In this manner, a single filter element can be used to filter multiple particle contaminants based on particle size alone. This can be used to define filters based on MERV test data [ASHRAE 1999a]. Only those species considered particle types and which you have selected to be contaminants will be filtered when you perform simulations.

Name: A unique name you want to use to identify the filter element.

Area: The face area and units of the filter element.

Depth: The depth of the filter element along the axis of airflow.

Density: The density of the filter media.

Area, Depth and Density are currently used to calculate the mass of the filter media which is only relevant for the Simple Gaseous Filter model.

Description: Use this field to provide a more detailed description of the filter element.

Edit Element Data: Select to define the specific properties of the filter element.

> **Filter Curve Data:** This is a list of *particle size* and *filter efficiency* data pairs used to define the filter curve. Once you have entered at least three data points, ContamW will attempt to

calculate a cubic spline fit to the points. If there are any errors in the spline fit procedure, you will be notified and the offending portion of the curve will be highlighted.

Use the **Particle Size** and **Efficiency** fields to set and modify the *Filter Curve Data* list along with the **Add**, **Replace** and **Delete** command buttons.

Particle Size: Use this to enter the particle size and units for which to define a *filter curve* data point. This corresponds to the *Mean Diameter* property of a species.

Efficiency: Use this to enter the *filter efficiency* for the associated particle size. Filter efficiencies must be between 0.0 and 1.0.

At the beginning of a simulation, ContamX will use the curve to establish a filter efficiency for each particle contaminant having a mean diameter in the range of the filter curve.

Simple Gaseous Filter

The *simple gaseous filter* element is made up of a set of species and associated efficiency vs. loading curves. Therefore, the efficiency is a function of the amount of contaminant "absorbed" by the filter. Each filter element can be associated with anywhere between one and the number of species that exist within the current project. Only those species that you have selected to be contaminants will be accounted for when you perform simulations.

Name: A unique name you want to use to identify the filter element.

Area: The face area and units of the filter element.

Depth: The depth of the filter element along the axis of airflow.

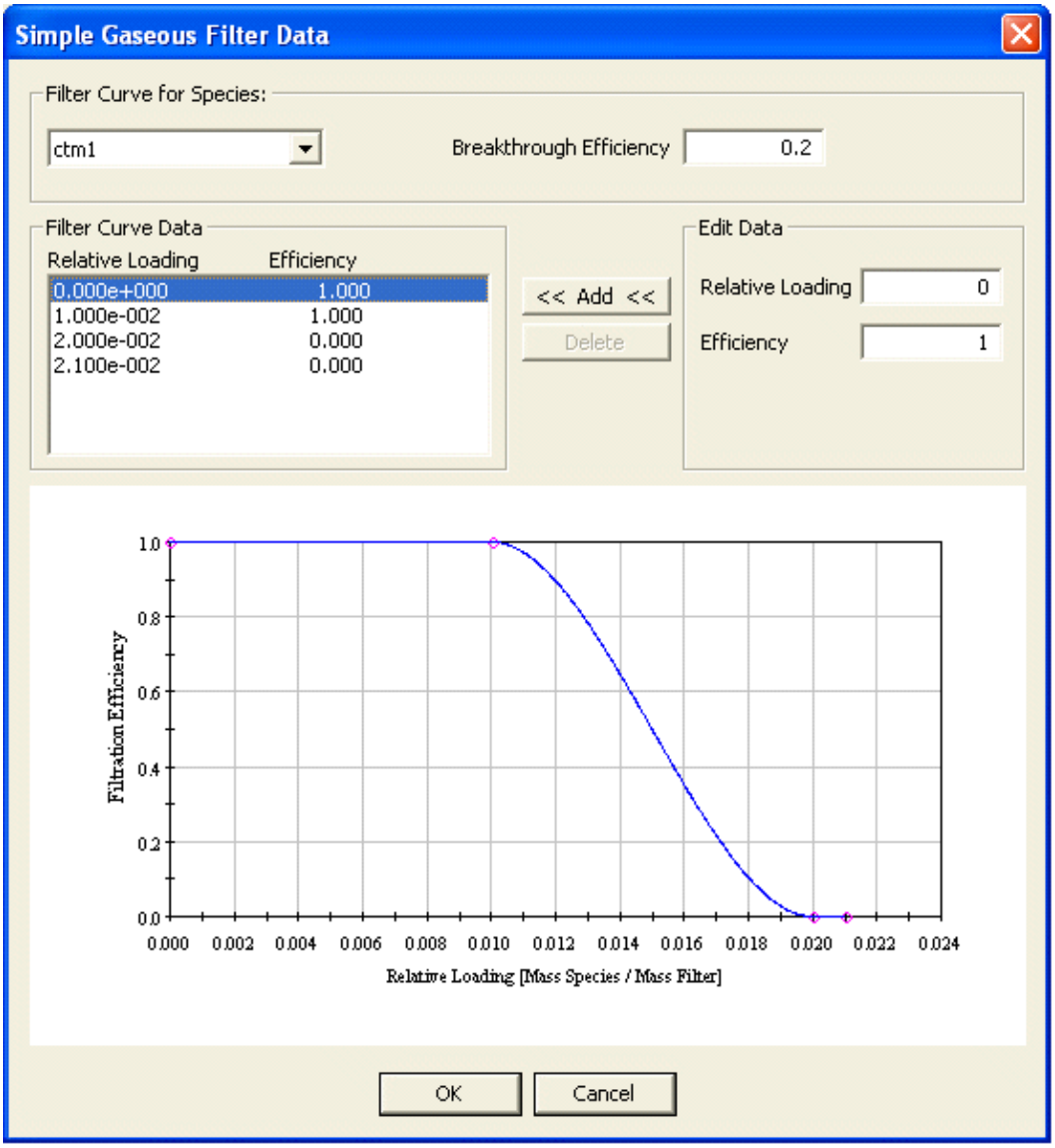
Density: The density of the filter media.

Area, Depth and Density are currently used to calculate the mass of the filter media which is in turn used to calculate filter loading.

Description: Use this field to provide a more detailed description of the filter element.

Edit Element Data: Select to define the specific properties of the filter element.

Filter Curve Data: This is a list of relative loading and filter efficiency data pairs used to define the filter curve for the currently selected species (see figure below). Once you have entered at least three data points, ContamW will attempt to calculate a cubic spline fit to the points. If there are any errors in the spline fit procedure, you will be notified and the offending portion of the curve will be highlighted.



Species: This is a list of all the *species* in either the current project or library file depending on whether you are editing project data or library data. Select the *species* for which you want to display/edit filter curve data.

Breakthrough Efficiency: Set the breakthrough efficiency for each species. This value will be used during simulation to report if and when the filter efficiency drops below this value. Breakthrough is reported to the *Contaminant Summary* simulation result file (see Results Files in the Working with Results section).

Filter Curve Data: This is the list of data points you create from which a *filter curve* is generated using the cubic spline fit method. A minimum of four data points is required before a curve will be generated. The space below will provide a plot of *Relative Loading* vs. *Filter Efficiency*. *Relative loading* is the ratio of Mass of contaminant accumulated by the filter to the Mass of the filter.

Use the **Relative Loading** and **Efficiency** data entry fields to set and modify the *Filter Curve Data* list along with the **Add** and **Delete** buttons.

Penn State UVGI Filter

The *Penn State UVGI* element implements an ultra-violet germicidal irradiation filter model based on work performed by the Pennsylvania State University [Lau 2009a, Lau 2009b]. The filter efficiency, i.e., UV output, is a function of temperature, velocity (airflow rate), the age of the filter relative and design UV output rate. The units provided below are those utilized internally by ContamX.

Name: A unique name you want to use to identify this filter element.

Area, Depth and Density are used to calculate the mass of the filter media, which is not used for this UVGI filter model, so these inputs can be ignored.

Description: Use this field to provide a more detailed description of the filter element.

Edit Element Data: Select to define the specific properties of this filter element. Units are provided in the default calculation units.

Design Survivability, S_{design}: Enter the desired survivability of the filter as a fraction between 0 and 1.0. The single pass filter inactivation efficiency is equal to 1 - S.

Design Velocity, u_{design} [m/s]**:** Enter the design maximum velocity expected through the filter flow region.

Design Organism-Specific Rate Constant, k_{design} [m^2/J]: Enter the design minimum microorganism-specific rate constant. This will be used to determine the design UV dose. You will also need to enter a value for each species that you want filtered by this UVGI filter element (see Species Properties).

Design Mass Flow Rate, F_{des} [kg/s]**:** Enter the design maximum airflow rate expected through the filter flow region. This will be used along with the Design Velocity to calculate the Cross-Sectional Flow Area.

Cross-sectional Flow Area, A [m^2]: $A = F_{design} / (\rho_{std}\, u_{design})$

Design Dose, $(It)_{design}$ [J/m^2]: $(It)_{design} = \ln(S_{design})/-k_{design}$

TU Constants: C0 through C4 are used to determine the output as a percentage of design, OUT_{TU}, based on current conditions including flow velocity and temperature.

Use Age Model: Select this option to have the output also determined as a function of lamp age.

Age Constants: K0 and K1 are the coefficients used to determine the percentage of output, OUT_{age}, based on the lamp age function.

Background

In its simplest form, survivability of an airborne microorganism exposed to UV-C radiation, *S*, presented in equation (uv-1), is approximated by a single-stage exponential decay equation [Lee and Bahnfleth 2013].

$$S = \frac{N_t}{N_0} = e^{-k(I\tau)} \qquad \text{(uv-1)}$$

Efficiency is then 1 minus the survivability.

$$\eta_{uvgi} = 1 - S = 1 - e^{-k(I\tau)} \tag{uv-2}$$

The UV dose is the irradiance over a period of time, $I\tau$. For design purposes, one might specify a design survivability, S_{design}, and a design minimum microorganism-specific rate constant, κ_{design}, to determine the design UV dose required:

$$(I\tau)_{design} = \frac{\ln(S_{design})}{-k_{design}} \tag{uv-3}$$

Any microorganism having an associated rate constant greater than or equal to k_{design} would have at most a survivability of S_{design}.

UV output is determined as a percentage of rated output using two relationships: one based on current conditions of the airflow around the lamps, i.e., temperature and velocity, and the other based on the age of the lamps [Lau 2009a, Lau 2009b]. These relationships are presented in equations (uv-4) and (uv-5), respectively. Temperatures for airflow paths and ducts are obtained from the upstream zone or junction respectively, based on the user-defined positive airflow direction.

$$OUT_{TU} = C_0 + C_1 T + C_2 U + C_3 T^2 + C_4 U^2 \tag{uv-4}$$

$$OUT_{age} = K_0 + K_1 \ln(t) \tag{uv-5}$$

Equations (uv-4) and (uv-5) are multiplied to provide the fractional UV output OUT_{uv1} provided in equation (uv-6). OUT_{age} is maintained at the default value of 1.0 until elapsed simulation time reaches t_{100} as defined by equation (uv-7).

$$OUT_{uv1} = OUT_{TU}\, OUT_{age} \tag{uv-6}$$

$$t_{100} = e^{\frac{100-K_0}{K_1}} \tag{uv-7}$$

During simulation the UV efficiency will be calculated based on the output relative to design conditions according to equation (uv-2) whereby $I\tau(T,U)$ will be determined by the following equation (uv-8) or as a function of airflow rate presented in equation (uv-9).

$$I\tau(T,U) = I\tau_{design}\, OUT_{uv1} \frac{U_{design}}{U} \tag{uv-8}$$

$$I\tau(T,U) = I\tau_{design}\, OUT_{uv1} \frac{Q_{design}}{Q} \tag{uv-9}$$

Super Filter

Use this element to combine multiple filter elements, in series, into a single filter.

Name: A unique name you want to use to identify the super filter element.

Description: Use this field to provide a more detailed description of the filter element.

Sub-Elements: You build this list of sub-elements from the list of *Available Elements* The order in which these elements appear in the list is important. They should be placed in the order in which air is to flow through the filter from top to bottom in the list. Use the Add, Delete and Move buttons to modify this list.

Available Elements: This is the list of fundamental filter elements, e.g., *constant efficiency*, *simple gaseous*, and *simple particle*, contained in the current project file. Use the *Edit Element* and *New Element* buttons to modify this list.

3.12 WORKING WITH KINETIC REACTIONS

CONTAM provides the ability to model chemical reactions among project contaminants within zones of a building including the implicit supply and return nodes of a simple air-handling system (See Air Handling System). Kinetic reactions in CONTAM are modeled as first-order exponential functions between species that contain the paired first-order reaction rate coefficients between the reactant and product contaminants. See Axley [1988] and Axley [1995] for a detailed presentations of the kinetic reaction simulation method employed by ContamX. The production or destruction of a contaminant α is given by

$$R_\alpha = \sum_\beta \left(K_{\alpha,\beta} \cdot C_\beta \right)$$

where

R_α rate of production or destruction of contaminant α

$K_{\alpha,\beta}$ reaction rate coefficients between contaminants α and β, in units of 1/s

C_β concentration of a reactant

CONTAM uses the convention that positive reaction coefficients produce an increase in species concentration and negative coefficients decrease species concentrations. The units of concentration are kg_species / kg_air (See Concentration Conversions).

Kinetic reactions can occur within zones of a building. To use kinetic reactions, you must have at least one species for the project. Kinetic reactions can be associated with one or more zones within a project. You can access the kinetic reaction data dialog box via the property sheets of the zones that provide a kinetic reaction definition section (i.e. zones and simple air-handling systems).

You can also access the Kinetic Reaction Data dialog box using the CONTAMW Data and Library Manager: Kinetic Reactions to create and edit local project data or to share kinetic reactions between CONTAM projects using contaminant libraries.

When you are defining building zones, you will be able to select a currently defined kinetic reaction to associate with it, define new, modify existing or import kinetic reactions from species-related CONTAM library files (i.e. LB0 library files). All kinetic reactions are defined and modified using the Kinetic Reaction Data dialog box as described in the following section.

Kinetic Reaction Data

Kinetic reaction elements are made up of source/product species pairs. You can define any number of pairs per element you desire. Only those species that you have selected to be contaminants will be accounted for when you perform simulations. These are the properties that describe a kinetic reaction.

Name: Enter a unique name you want to use to identify the reaction matrix. You will use this name to associate this element with zones within the project.

Description: Use this to provide a more detailed description of the kinetic reaction.

Reactant→Product Pairs: This is a list of the reactant/product pairs and their associated reaction coefficients that are currently defined for the current kinetic reaction element.

Edit Reactant→Product Pair: Use this section to edit the highlighted Reactant→Product pair or create new ones. Use the Add, Replace and Delete buttons accordingly.

Reactant Species: This is a list of all the species in either the current project or library file depending on how this dialog box was accessed. Select a species to be the reactant in the R→P Pair.

Product Species: This is a list of all the species in either the current project or library file depending on how this dialog box was accessed. Select a species to be the product in the R→P Pair.

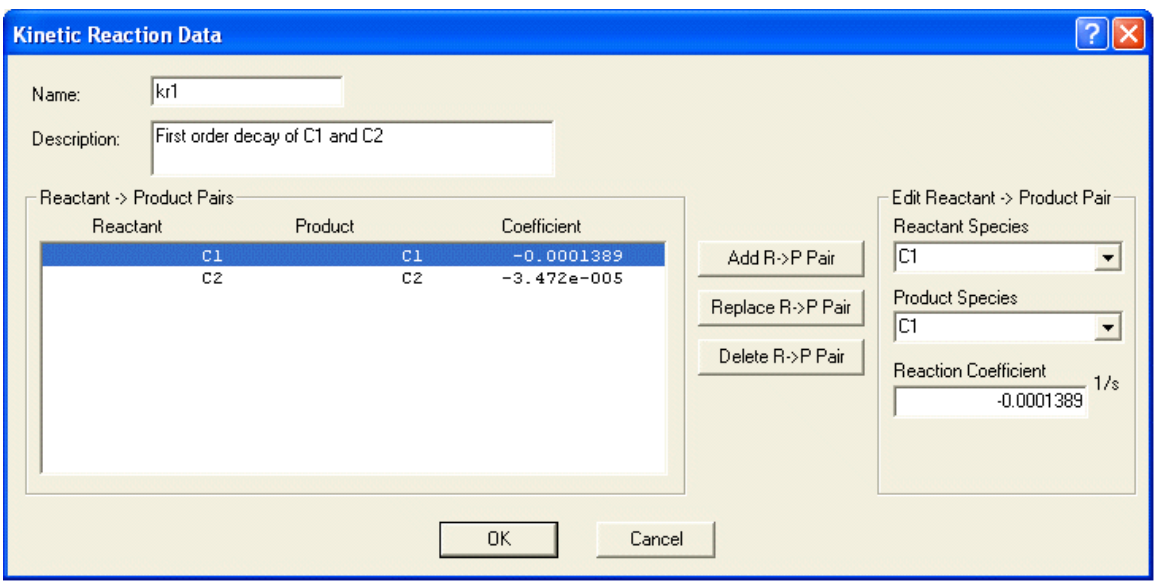
Reaction Coefficient: Use this edit field to set the reaction coefficient for the currently highlighted R→P Pair. The units of the coefficient are s^{-1}.

Examples

The following are examples of kinetic reactions as implemented within CONTAMW.

Kinetic Reaction Example 1

To simulate the first-order decay of a species, you would set the coefficient of the Reactant→Product Pair to the negative of the reaction rate in units of 1/s.



This figure shows kinetic reaction coefficients for two species reactions that will yield exponential decays of each species according to the following equation:

$$C_j(t) = C_i(0)e^{K_{ji}t}$$

Where,

$C_j(t)$ = concentration of product contaminant j at time t

$C_i(0)$ = initial concentration of reactant contaminant i at time 0

K_{ji} = reaction rate coefficient between product j and reactant i (time constant)

In the above example, the reaction rate coefficients are as follows:

K_{11} = -0.0001388/s
K_{22} = -0.00003472/s

If the reaction is the only means of species removal, i.e., there is no dilution due to airflow, then species C1 would decay at a rate of

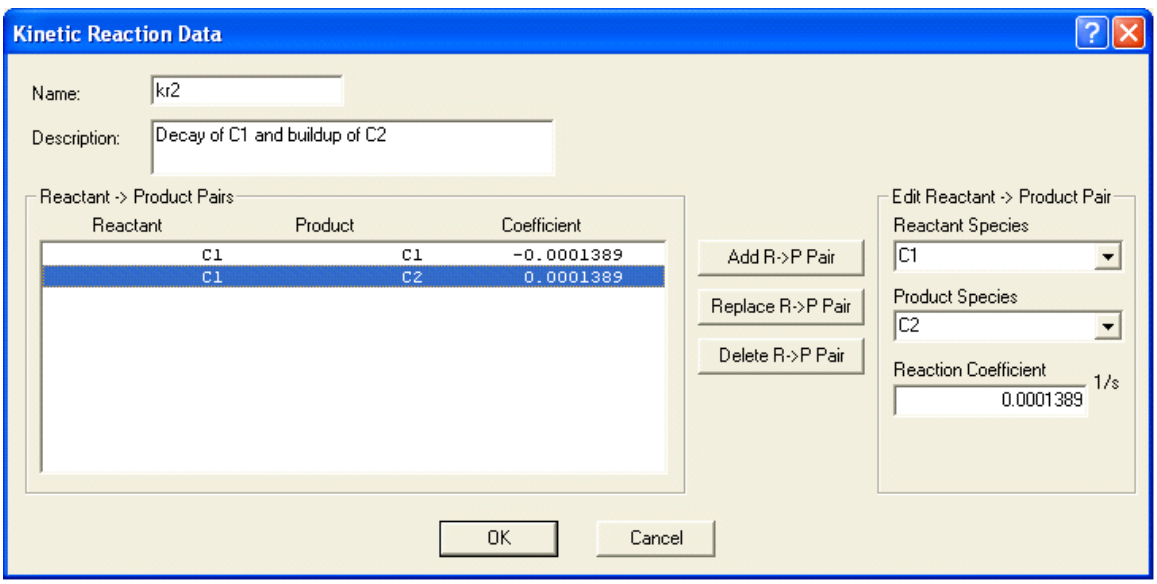
0.0001388 * 3600 = 0.5/h,

and C2 would decay at a rate of

0.00003472 * 3600 = 0.125/h.

Kinetic Reaction Example 2

This example illustrates the first-order decay of species C1 (reactant and product) and the first order build-up of another species C2 (product) based on C1(reactant).



This figure above shows the parameters for the "kr1" kinetic reaction element. This element provides for two reactions that will yield an exponential decay of contaminant C1 according to the equation

$$C_1(t) = C_1(0)e^{K_{11}t},$$

and the build-up of contaminant C2 according to

$$C_2(t) = C_1(0)\left(1 - e^{K_{21}t}\right)$$

In the above example, the reaction rate coefficients are as follows:

K_{11} = -0.0001389/s
K_{21} = 0.0001389/s

Note that both the decay and build-up are dependent on the species of C1.

Kinetic Reaction Example 3

You can also use the kinetic reaction to simulate a consecutive first-order reaction series. (The radioactive decay chain of Radon gas is an example of such a reaction.)

A two-step consecutive reaction series involving a single reactant at each step is indicated as

C1 → C2
C2 → products

You would model this reaction chain by inputing coefficients based on the half-life of the radioactive reactants. Convert half-life $\tau_{1/2}$ to a first-order reaction coefficient, K_{21}, using the following equation:

$$K = \ln(0.5) / \tau_{1/2}$$

In CONTAM, you would require two species C1 and C2 (if interested in contaminant C2) and three reactant/product pairs have the following reaction coefficients characteristics:

K_{11} would have negative non-zero values (decay of C1)

K_{21} would have a non-zero positive value (build-up of C2 from C1)

K_{22} would have negative non-zero values (decay of C2)

3.13 WORKING WITH OCCUPANT EXPOSURE

CONTAM provides you with the ability to simulate the existence of *occupants* within a building. You can simulate the movement of occupants throughout the building, allow the occupant to leave the building, and determine the contaminant exposure of each occupant. Occupants can also generate contaminants within a building, so you can represent a contaminant source that moves through the building using an occupant.

Occupant exposure is determined by integrating the contaminant concentration to which the occupant is exposed over a period of time.

$$E = \int_{t_1}^{t_2} C(t)dt$$

CONTAM presents results in terms of $E/\Delta t$. Where Δt is the user-defined *output time step* established for a transient simulation (See Results Files in the Working with Results section). From these results you can calculate a dosage as desired.

To implement an occupant, you must first define at least one species. You then place an *exposure icon* (shown below) on the SketchPad. Once you place the exposure icon on the SketchPad, you can define its characteristics, move, copy, and delete it. You must then define an occupant schedule for the exposure icon. The details of drawing, defining and modifying occupants are described in the following sections.

Occupant exposure icon	

3.13.1 Creating Occupants

You create an occupant by placing an exposure icon on the SketchPad (See Drawing Building Component Icons). You can place them in any blank cell of the SketchPad. The "Exposure" menu selection will be disabled (grayed out) if you pop-up on a location where ContamW does not allow the icon to be placed.

3.13.2 Deleting Occupants

You delete exposure icons using the icon deletion procedure (See Deleting Building Component Icons). Deleting occupant exposure icons from the SketchPad deletes the occupant data associated with the icon, but it does not delete any occupant-related schedules that you might have created. These schedules can only be deleted using the CONTAMW Library Manager, accessible through the **Data** menu for each type of data element.

3.13.3 Modifying Occupants

To modify the parameters of an occupant, you use the icon definition procedure (See Defining Building Component Icons) to display its properties and make the desired changes. You can also move and copy occupants.

Moving Occupant Exposure Icons

Once you define an occupant, you can move the icon using the **Cut** and **Paste** functions of the **Edit** menu. You can move the occupant exposure icon anywhere within the level upon which it is currently located.

Copying Occupant Exposure Icons

Once you define an occupant, you can copy the icon using the **Copy** and **Paste** functions of the **Edit** menu. You can copy an occupant to any blank cell on any level of a project.

3.13.4 Defining Occupants

After you draw an exposure icon on the SketchPad you must define it using the icon definition procedure (See Defining Building Component Icons) to display and edit the properties. This will display the "Occupant Exposure Properties" dialog box. Detailed descriptions of all occupant exposure properties are given in the following Occupant Properties section. Once you have defined the properties, the icon will be displayed in black. From now on, you use this icon to access the properties of the occupant.

3.13.5 Occupant Properties

This section provides detailed descriptions of the specific occupant properties including occupancy schedule, i.e., location within the building over time and the occupant-associated contaminant generation. Each of the following subsections are the context-sensitive help topics that you can access by pressing **F1** when working with property pages of the "Occupant Exposure Properties" property sheet.

Occupant – Occupancy Schedule Properties

Occupant Number: This is the number that appears in the status bar when you highlight an icon. ContamW automatically assigns this number to each exposure icon once it is defined. This number could change as you add and remove occupants to or from a project. Each time you save a project, the occupant numbers are reassigned. Numbers are assigned beginning on the top level, starting from the upper-left corner of the SketchPad, moving left-to-right and top-to-bottom of the SketchPad.

Occupant Week Schedule: Use the Occupancy Week Schedule to define the movement of this occupant within the building (see Working with Schedules).

Occupant – Contaminant Generation Properties

Multiplier: Use the multiplier to modify the generation rate of an occupant. You could define the contaminant generation rate(s) for a single occupant and use the multiplier to represent a generation rate of multiple occupants.

Generate Contaminants: This checkbox is used to turn on and off the contaminant generation of the occupant. If the box is checked, contaminants will be generated. This is useful for contaminant generation such as that of carbon dioxide due to human metabolic activity.

Generation Data: A list of available contaminants is displayed in the text box provided. You can highlight these contaminants and select the generation rate for each in the edit box below the contaminant list.

Generation Schedule: A new inhalation schedule is defined by pressing the "New Schedule" button and following the directions for editing a Week Schedule.

> **Description:** Field that shows a more detailed description of the specific schedule if entered by the user. This description can be modified by pressing the edit schedule button next to the description field and then changing the description field for the schedule.

Name: This drop down box contains the names of all previously created schedules. The schedule selected from this box will then define the generation schedule for the exposure element being viewed.

CVF/DVF Control: Values from either a CVF or DVF file can be used in place of the above Schedule (read about CVF and DVF controls in the Control Element Types section). This enables using these control element types without having to draw/define an associated control network. The associated radio buttons "Use Neither", "Use CVF" and "Use DVF" are used to select the desired control method. Select the heading name from the associated "Name" drop down box that will display the headings of either the CVF or DVF file as per the aforementioned radio button selection.

Control Status: This will display the current method of control: Schedule, direct CVF, direct DVF or Control Network. If an actuator of a control network on the SketchPad is associated with the scheduled item then it will take precedence over the Schedule and CVF/DVF Control. If either the "Use CVF" or "Use DVF" radio button is selected then CVF or DVF will take precedence over an associated schedule.

3.14 Working with Data and Libraries

CONTAM enables you to share certain types of data between building components contained within a project as well as between projects. The sharing of data within a project (locally) is accomplished by allowing you to associate multiple building components with *data elements* defined within a project. Sharing data between projects is accomplished by exporting data elements from one project into CONTAM library files and then retrieving the data elements into another project from the library files. The following table lists the different types of data elements implemented by CONTAM. All of the data elements can be shared within a project, and all but occupancy schedules can be shared between projects.

Library/Local Data Elements
Contaminant-related (LB0)
Species
Source/Sink Elements
Contaminant Filters
Kinetic Reactions
Schedules (LB1)
Wind Pressure Profiles (LB2)
Airflow Elements (LB3)
Ductflow Elements (LB4)
Control Super Elements (LB5)
Local-Only Data Elements
Occupancy Schedules

Data Elements

You create and modify data elements according to the procedures given throughout the "Using CONTAM" section of this manual for the specific data elements. Once you have created a data element, you can associate it with the relevant building components as you create them. You do this by selecting the desired element from the local list that ContamW presents to you when you are editing the properties of a particular building component. To share data elements between project files, you must utilize the CONTAMW Library Manager. You must also use the CONTAMW Library Manager to delete data elements from either the local project or the library. To delete local data elements, they must not be referenced by any building components.

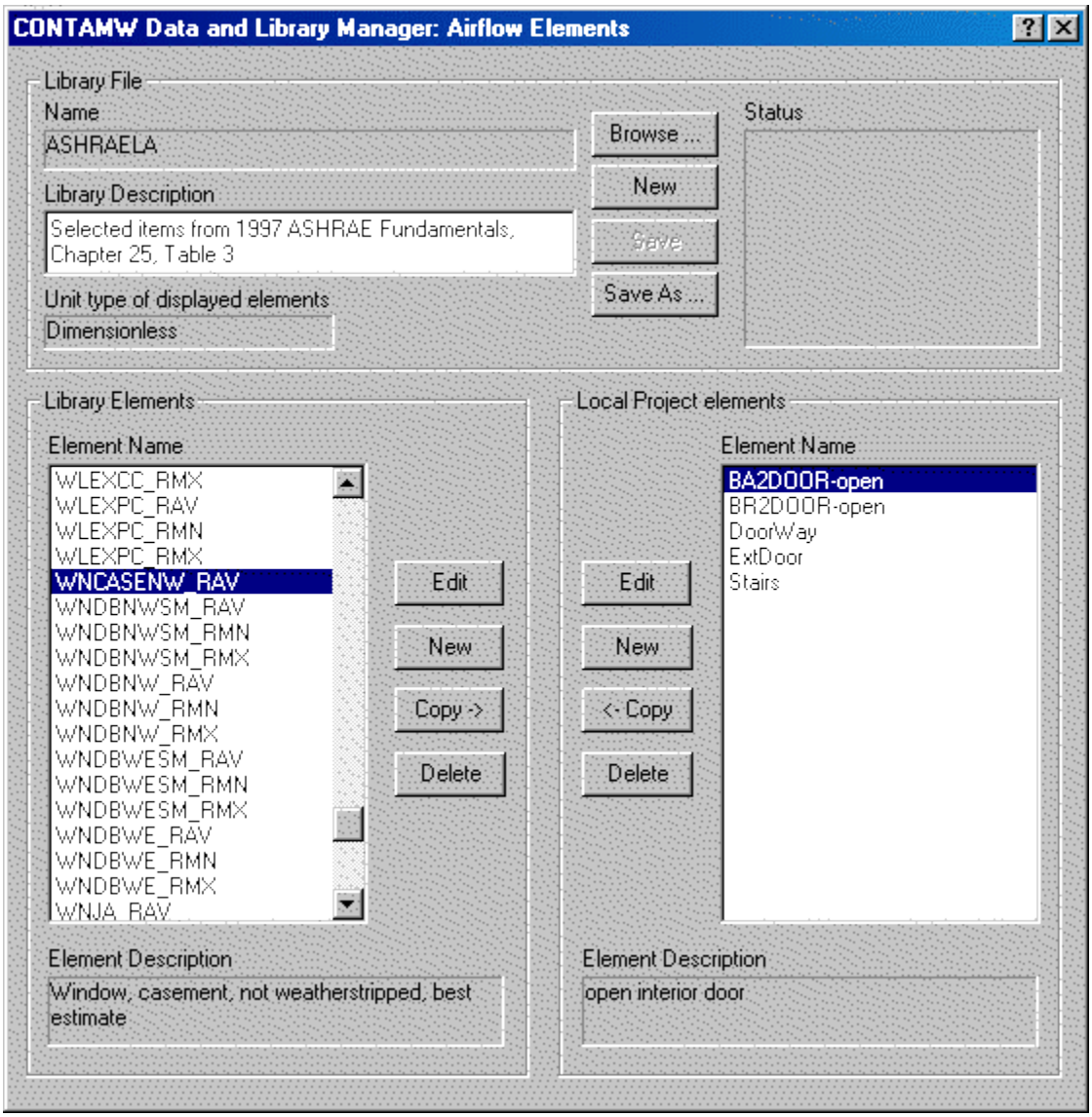
Library Files

CONTAM Libraries are files that contain the descriptions of sharable data elements. Each type of sharable data element is stored in a different library file. You can create as many of each type of library file that you want, and store them anywhere within the file structure of your computer. Each type of library file is associated with a different three-character extension as shown in the table above.

CONTAMW Library Manager

The CONTAMW Library Manager, shown in the following figure, is a dialog box that is displayed when you access data elements from the **Data** menu or by clicking on a "Library…"

button when editing the properties of one of the building components that utilizes library data elements. This dialog box is divided into three sections: Library File, Library Elements and Local Project Elements sections. The Library File section displays the name of the currently displayed library file and allows you to save, rename and open new and existing library files. The Library Elements section displays the data elements contained in the library file that is currently listed in the Library File section. The Local Project Elements section displays the data elements that are contained in the current project file.



3.14.1 Opening Libraries

If a library file is currently opened, the name of the file will be displayed in the "Name" field of the "Library File" section of the Library Manager. You can open a CONTAM library file using the "Browse…" button of the Library Manager. This will activate the File Open dialog box typical of Windows applications. This dialog box is set to display only those files having the three-character file extension associated with the type of data elements that you are currently

working with. Follow the typical procedure for opening a file using the Windows operating system.

CONTAM 2.0 library files are of a different format than the 1.0 version. ContamW will allow you to open a 1.0 version file and you will then be required to give the file a name when saving the converted file. It is suggested that you save the file under a different name from that of the 1.0 formatted library file. The Open dialog box will display both the file *Version* information and the Library file *description*.

3.14.2 Creating Libraries

Using the CONTAMW Library Manager, you create new libraries one of two ways. You can either save an existing library under a new name using the "Save As…" button, or use the "New" button of the Library File section of the CONTAMW Library Manager dialog box. The "Save As…" method makes a copy of the currently displayed library file under a new name, but maintains the original file. The "New" method clears the data currently displayed and allows you to start with an empty library file.

3.14.3 Deleting Libraries

You must use the file deletion commands of your operating system to delete CONTAM library files.

3.14.4 Modifying Libraries

You modify CONTAM library files using the CONTAMW Library Manager. The CONTAMW Library Manager allows you to create new data elements within the library, copy data elements from the local project into the library, modify library data elements, and delete data elements from the library.

Creating Library Data Elements

You use the "New" button contained within the "Library Elements" section of the CONTAMW Library Manager to create a new data element. You can only create the type of data element that is currently being displayed by the Library Manager (e.g. airflow elements, ductflow elements, schedules, etc.). The "New" button will activate the ContamW dialog box associated with creating the type of data element that you are currently working with.

Species must be available in order to create new contaminant-related elements such as Sources/Sinks, Filters and Kinetic reactions.

Copying Library Data Elements (between the Project and Library)

You can either copy data element from a library to the local project or from the local project to the library. All copying is done between the local project and the library file that is currently indicated in the "Name" box of the Library File section of the CONTAMW Library Manager. If the name box is blank, then you are dealing with a new/unnamed library file. To copy data elements, you use the "Copy →" and "← Copy" buttons. These copy buttons indicate the direction of the copy operation. If an element exists in both the library file and the local project, then the copy buttons will be disabled to prevent you from overwriting elements with the same name. You may rename local data elements without affecting their associations with building components that refer to them in the local project.

Some elements rely on other types of elements for proper implementation, e.g., source/sinks require an associated species and week schedules require day schedules. If you copy one of these elements that requires a sub element, ContamW will prompt you to replace existing elements of the same name if they already exist within the destination file.

Modifying Library Data Elements

To modify a library data element, you must first highlight the Element Name within the list of elements in the Library Elements section of the CONTAMW Library Manager. You then click on the "Edit" button next to the list. This will activate the ContamW dialog box associated with displaying the properties of the type of data element with which you are currently working. You then make the desired changes and click the "OK" button to make the changes take affect.

Deleting Library Data Elements

To delete a library data element, you highlight the Element Name in the Library Elements section of the CONTAMW Library Manager then click on the "Delete" button next to the list. You will be prompted to confirm the deletion, and the data element will no longer be displayed in the list of Library Elements. However, the deletion will not be affected until you save the library file. You can retrieve the previously deleted data elements by reopening the library file as long as you don't save the file after deleting the elements.

Deleting data elements that have associated sub-elements will not delete the sub-elements. You must delete the sub-elements using a separate Library Manager operation.

3.14.5 Modifying Local Data

You modify local lists of data elements using the CONTAMW Library Manager. The Library Manager allows you to create new local data elements, copy data elements from the library into the local project, modify local data elements, and delete local data elements.

Creating Local Data Elements

You use the "New" button contained within the "Project Elements" section of the Library Manager to create a new data element. You can only create the type of data element that is currently being displayed by the Library Manager (e.g. airflow elements, ductflow elements, schedules, etc.). The "New" button will activate the ContamW dialog box associated with creating the type of data element that you are currently working with.

Species must be available in order to create new contaminant-related elements such as Sources/Sinks, Filters and Kinetic reactions.

Copying Local Data Elements (between the Project and Library)

See Copying Library Data Elements.

Modifying Local Data Elements

The Library Manager provides you with another means of accessing local project elements. This is in addition to the method of accessing building component properties presented throughout the Using CONTAM section of this manual. You must first highlight the Element Name within the list of elements in the Project Elements section of the Library Manager. You then click on the "Edit" button next to the list. This will activate the ContamW dialog box associated with displaying the properties of the type of data element that you are currently working with. You then make the desired changes and click the "OK" button to make the changes take affect.

Copying Data Elements (within a Project)

You can use the Library Manager to create elements that are similar to existing elements but have a different name. To do this you copy a project element to the library (you don't have to have an existing library file open), give the library element a new name, make the modifications you desire and copy the element back into your project.

Deleting Local Data Elements

You can only delete local data elements if they are not referenced by any building components within the project. To delete a project data element, you highlight the Element Name in the Project Elements section of the CONTAMW Library Manager then click on the "Delete" button next to the list. The "Delete" button will only be enabled if the currently highlighted element is not referenced within the project. You will be prompted to confirm the deletion, and the data element will no longer be displayed in the list of Project Elements. However, the deletion will not be affected until you save the project file. You can retrieve the previously deleted data elements by reopening the project file as long as you don't save the file after deleting the elements.

Deleting data elements that have associated sub-elements will not delete the sub-elements. You must delete the sub-elements using a separate Library Manager operation.

3.15 WORKING WITH WEATHER AND WIND

CONTAM enables you to incorporate the effects of weather on a building. Weather parameters include ambient temperature, barometric pressure, wind speed, wind direction and also outdoor contaminant levels. The type of weather data you require for a simulation depends on the type of simulation you are performing, whether you want to account for the effects of wind, and whether or not you are simulating contaminants. If you are performing a steady-state airflow simulation, you will only need steady state weather and wind data. If you are performing a transient airflow simulation, you can use either steady state or transient weather data. Transient weather data is implemented through the use of weather files or WTH files. If you are performing transient contaminant simulations, you can use either steady-state or transient ambient contaminant data as well. Transient ambient contaminant data is implemented via the use of ambient contaminant files or CTM files (See Contaminant Files).

NOTE: Both the WTH and CTM files assume a single set of values for the ambient zone at any given point in time, i.e., there is no spacial variation in either the pressure or contaminant fields. However, CONTAM also allows the use of transient wind pressure and contaminant files (WPC file) with which you can account for spacial variation in both the pressures and contaminant concentrations over the surface of a building envelope (See Working with WPC Files).

Steady State Weather

When you use steady state weather data, ContamX keeps the ambient temperature, barometric pressure, wind speed and direction and relative humidity constant during a simulation. Detailed descriptions of these parameters are given in the following sections.

Transient Weather

You can use transient weather data to simulate the changing outdoor weather and wind conditions when performing a transient simulation. Transient weather data is stored in a weather file that you must create according to a special format as specified under the Weather File Format section that follows. Weather files contain ambient temperature, pressure, wind speed and direction and humidity ratio for discrete intervals of each day for at least one day and up to one year. If the intervals between data in the weather file don't match the simulation time step, ContamX will linearly interpolate the weather data to obtain a value at the required timestep.

Wind

Wind pressure can be a significant driving force for air infiltration through a building envelope. It is a function of wind speed, wind direction, building configuration, and local terrain effects. CONTAM enables you to account for the effects of wind pressure on flow paths through the building envelope (external airflow paths). You define how you want to account for wind effect for each external flow path on a case-by-case basis. In ContamW, you can either choose to ignore the effects of wind on an envelope penetration, define wind pressure to be constant for each envelope penetration, or implement variable wind pressures for envelope penetrations.

CONTAM provides a general approach to handling the variable effects of wind on the building envelope. This approach requires you to provide CONTAM with information related to the determination of a local wind pressure coefficient for the building surface.

For a general introduction to the effects of wind pressure on buildings, see Chapter 16, Air flow Around Buildings, in the 2005 ASHRAE Fundamentals Handbook [ASHRAE 2005]. The following is an overview of how CONTAM handles these effects.

The equation for wind pressure on the building surface is

$$P_w = \frac{\rho V_H^2}{2} C_p$$

where

V_H Approach wind speed at the upwind wall height (usually the height of the building)

C_p Wind pressure coefficient

The wind pressure coefficient can be further generalized in terms of a local terrain effects coefficient and the direction of the wind relative to the wall under consideration. The following equation is that used by CONTAM when calculating wind pressures on the building.

$$P_w = \frac{\rho V_{met}^2}{2} C_h f(\theta)$$

where

ρ ambient air density

V_{met} wind speed measured at meteorological station

C_h wind speed modifier coefficient accounting for terrain and elevation effects

$f(\theta)$ coefficient that is a function of the relative wind direction. CONTAM refers to this function as the *wind pressure profile*.

The relative wind direction is given by

$$\theta \equiv \theta_w - \theta_s$$

where

θ_w wind azimuth angle (N = 0°, E = 90°, etc.), and

θ_s surface azimuth angle.

$\rho V^2_{met} / 2$ in the above equation is computed by ContamX from the steady state or weather file temperature, pressure and wind speed data.

C_p, the pressure coefficient used by ASHRAE, is equivalent to $f(\theta)$ in the CONTAM formulation. In ContamW, you must define the function $f(\theta)$ in the form of *wind pressure profiles* as explained later. See page 27.5 of the 2005 ASHRAE Fundamentals Handbook [ASHRAE 2005] for details relating to pressure coefficient data. There are some examples of wind pressure profiles available in the form of CONTAM library files on the NIST website (http://www.bfrl.nist.gov/IAQanalysis/CONTAM/libraries.htm).

The wind pressure modifier, C_h, accounts for the difference between V_{met} and V_H. The value for C_h will typically be constant for all openings on a given building. However, ContamW allows you to define a default value that will be used for each flow path connected to the ambient (See Wind Properties) or to set this value individually for each of these paths. ContamW uses the following formula to compute the value of C_h:

$$C_h = \frac{V_H^2}{V_{met}^2} = A_o^2 \left(\frac{H}{H_{ref}} \right)^{2a} \tag{1}$$

where H is the wall height and A_o and a depend on the terrain around the building [ASHRAE 1993, p 14.3]:

Terrain Type	Coefficient (A_o)	Exponent (a)
Urban	0.35	0.40
Suburban	0.60	0.28
Airport	1.00	0.15

The wind speed, V_{met}, is usually measured at an airport (open terrain) at an elevation, H_{met} = 10m, above the ground. The wind speed, V_o, at that same elevation at the building site is given by

$$V_o = A_o V$$

The wind speed, V_H, at the top of the wall, elevation H, is then given by

$$V_H = V_o \left(\frac{H}{H_{met}} \right)^a$$

An updated method, from that presented in ASHRAE 1993, of accounting for local terrain effects is presented Chapter 16 of ASHRAE 2005. Using this method, V_H is calculated according to the following equation:

$$V_H = V_{met} \left(\frac{\delta_{met}}{H_{met}} \right)^{a_{met}} \left(\frac{H}{\delta} \right)^a$$

where

δ_{met} Wind boundary layer thickness for the meteorological station

δ Wind boundary layer thickness for the local building terrain

a_{met} Wind boundary layer exponent for the meteorological station

a Wind boundary layer exponent for the local building terrain

Table 1 in Chapter 16 of ASHRAE 2005 contains values for the above parameters. ContamW calculates C_h according to equation (1) above and provides for the input of the *local terrain constant*, A_o, but not δ. Therefore, in order to implement the updated method, you must adjust the value of A_o according to the following equation:

$$A_o = \left(\frac{\delta_{met}}{H_{met}} \right)^{a_{met}} \left(\frac{H_{met}}{\delta} \right)^a$$

3.15.1 Defining Weather and Wind

You define weather that you want CONTAM to use based on the type of simulation you want to perform. You will either define steady state weather or transient weather. It is best to perform a set of steady state simulations prior to performing transient simulations. This may enable you to better identify any problems that may arise from working with transient weather. ContamW also provides a means to verify that you have your wind related data input correctly (See Checking Wind Pressure Data).

Defining Steady State Weather and Wind

To define the steady state weather and wind data you use the **Edit Weather Data…** selection of the **Weather** menu or the **F7** keyboard shortcut. This will display the "Weather and Wind Parameters" property sheet. The property sheet contains four different pages of parameters including: Weather, Wind, Location and Wind Pressure Display. Detailed descriptions of all wind and weather properties are given in the Weather and Wind Properties section of this manual.

You use the Weather page to define steady state weather and wind data. The Location page allows you to input the altitude of the building site which ContamW will use to determine a default barometric pressure if you do not know what it should be for a given building site. Default values of these two pages will provide conditions of no wind, 20 °C and a barometric pressure of 1 atmosphere (approximately 101 kPa).

The Wind page parameters allow you to enter the prevailing wind direction and other values used by ContamW to determine a wind speed modifier. The wind speed modifier is determined from the local terrain constant and the velocity profile exponent. Default values are provided for suburban terrain.

Checking Wind Pressure Data

ContamW enables you to verify wind information visually on the SketchPad using the **Wind Pressure** and **Wind Pressure Type** selections of the **View** menu. When reviewing this display, you should check if the direction of flow makes sense based on the wind data provided on the Wind Pressure Display property page (WP Display) or contained in the WTH and WPC file (see Working with WPC Files). When reviewing WTH and WPC file pressures, you can use the **Results Navigation** controls under the **View** menu to step through the data in the WPC file. The date and time of the corresponding data from either file will be displayed in the status bar.

When you choose to display wind pressure results, colored lines (Pres Results color) will appear on the SketchPad. Wind pressure will only be displayed for those airflow paths and duct terminals connected to the ambient zone, having the Wind Pressure Option set to Variable or Constant or, if using a WPC file, a pressure specified in the WPC file. The wind pressure display will reveal the relative magnitude of the pressure drop and the direction of airflow across the openings in the building envelope. You can move the caret to each of the airflow paths for which a wind pressure is being displayed to show the wind pressure, azimuth angle and elevation of the airflow path in the ContamW status bar. When the caret is placed in the ambient zone, the wind speed and direction will be displayed in the status bar if the wind pressure type is set to either WP Display or WTH.

Defining Transient Weather and Wind

In order to implement transient weather and wind, you must create a CONTAM-compatible weather file. You must create the weather file external to CONTAM. NIST provides a utility program, WEATHER 2.0, to create CONTAM-compatible weather files.

WEATHER 2.0 enables you to convert existing CONTAM 1.0 weather files to the 2.0 format weather and ambient contaminant files. You can also convert TMY2 and EnergyPlus (.EPW) weather files to the CONTAM 2.0 weather file format and create weather file templates into which you can place your own data (e.g., experimental data). Both weather and ambient contaminant files are ASCII text files that can be created and edited using spreadsheet software according to the formats provided within this manual.

Weather File Format

The CONTAM weather file is a tab-delimited ASCII text file. These files are easily created, imported and modified using typical spreadsheet applications. They can be saved as tab-delimited text files for use with CONTAM. It is described below because it may be necessary for the you to create a weather file from sources not already provided. In the listing below `I1` indicates a character string, `I2` indicates a short integer and `R4` indicates a four-byte real number. Individual lines of data are separated by dashes (these dashes are only shown here for format presentation purposes and should not be included in the actual files). Each day must begin with time 00:00:00 and end with time 24:00:00. The time interval between 00:00:00 and 24:00:00 can be either regular or irregular – ContamX will interpolate as necessary. Comments are allowed and are indicated by an "!" (exclamation point). Anything that appears on a line following a comment indicator is ignored by the weather file reader. Comments are provided by the weather file creation software (Weather 2.0) that show header information for the data in the file.

```
WeatherFile ContamW 2.0 ! file and version identification
-----
I1 descr[] ! weather file description
-----
I1 start _date ! day of year (1/1 – 12/31)
I1 end_date ! day of year (1/1 – 12/31)
-----
!Date DofW Dtype DST Tgrnd
-----
! for each day: start_date – end_date
 I1 date ! date (1/1 – 12/31)
 I2 dayofwk ! day of week Sun – Sat (1 - 7)
 I2 daytype ! type of day for schedule reference (1 – 12)
 I2 DST ! daylight savings time indicator (0 or 1)
 R4 Tground ! ground temperature [K]
-----
!Date Time Ta Pb Ws Wd Hr Ith Idn Ts Rn Sn
-----
! for each day: start_date – end_date
! for each time: 00:00:00 - 24:00:00 increment may be variable
```

```
 I1 date ! date (1/1 - 12/31)
 I1 time ! time of day (00:00:00 - 24:00:00)
 R4 tmpambt ! ambient temperature [K]
 R4 barpres ! barom. pressure(Pa) NOT corrected to sea level
 R4 windspd ! wind speed [m/s]
 R4 winddir ! wind direction [deg]: 0=N, 90=E, 180=S, ...
 R4 humratio ! humidity ratio [g H20/kg dry-air]
 R4 solhtot ! total solar flux on horizontal surface [W/m^2]
 R4 solhdif ! diffuse solar flux on horizontal surface [W/m^2]
 R4 tskyeff ! effective sky temperature [K]
 I2 rain ! rain indicator 0 or 1
 I2 snow ! snow indicator 0 or 1
-----
```

The Tground, solar, sky, rain and snow values are not used in CONTAM. They are reserved for thermal analysis, if it is added to the program.

Editing Weather Files

You can import weather files into a spreadsheet program for editing. However you must make sure that the date format is compatible with CONTAM when you save the file. One well-known spreadsheet program will convert the CONTAM date format of mm/dd into the mmm-dd form. For example the CONTAM date format for January 1 is "1/1", but the spreadsheet program will convert it to "Jan-01" unless you "force" it to import the first column of data – the one that contains date information – as text. Another option is to accept the default importing option then change the cell format for each date to the CONTAM format mm/dd prior to saving the file as tab-delimitted.

3.15.2 Weather and Wind Properties

This section provides detailed descriptions of the specific steady state weather properties. Each of the following subsections are the context-sensitive help topics that you can access by pressing **F1** when working with property pages of the "Weather and Wind Properties" property sheet.

Weather and Wind – Wind Properties

The wall height, terrain and exponent are used to calculate the wind speed modifier.

Relative North: The direction to relative north will be used relating wall angles on the SketchPad to true north for determining wind direction effects. Enter the clockwise angle from vertical on the ContamW SketchPad to true north.

Roof or Wall Height: Enter the height of the upwind wall of the building. This is the value of H in the equations presented in the section Working with Weather and Wind. Entering a value of zero will reduce the wind speed modifier to zero. This will cause computed wind pressures to be zero no matter what the wind speed, unless you override the wind pressure coefficients of exterior airflow paths. Figure 4 in Chapter 16 of the 2005 ASHRAE Fundamentals Handbook [ASHRAE 2005] shows relative wind pressures on the face of a building based on the local wind pressure at the top of the wall.

Terrain & Exponent: Enter the values that describe the wind speed profile for the type of region in which your building is located. Figure 4 in Chapter 14 of the 1993 ASHRAE

Fundamentals Handbook [ASHRAE 1993] shows terrain constants and velocity profile exponents for representative areas.

An updated method, from that presented in ASHRAE 1993, of accounting for local terrain effects is presented Chapter 16 of ASHRAE 2005. Refer to Working with Weather and Wind for details on how to implement this method within CONTAM.

Modifier: This is the wind speed modifier, C_h, calculated by ContamW based on the Wall Height, Terrain and Exponent values above. This value will be the default provided for each envelope airflow path that you create on the SketchPad and for which you select the variable wind pressure option. See section Working with Weather and Wind for a more detailed explanation of this term.

Weather and Wind – Weather Properties

The following constant values are used by CONTAM when you specify the use of steady state weather conditions when performing simulations (See Working with Simulations).

Temperature: ContamX uses the ambient temperature when calculating outside air density and building stack effects.

Absolute Pressure: Absolute barometric pressure (not corrected to sea level). If you don't know this value, you can enter the altitude of the building site on the Location Properties page to have ContamW calculate the barometric pressure for you.

Relative Humidity: Relative humidity as a fraction, i.e., between 0.0 and 1.0. This value will be used along with Ambient Temperature and Pressure to calculate the Humidity Ratio and Mass Fraction of H2O.

Humidity Ratio: The mass of water vapor per mass of air (excluding water vapor) calculated based on the Temperature, Pressure and Relative Humidity.

Humidity Ratio = Mass Fraction / (1 – Mass Fraction)

Mass Fraction (H2O): The mass of water vapor per mass of air (including water vapor). These are the default units of contaminants in CONTAM. This value is provided so you can use it as the default contaminant concentration of H2O.

Wind Speed: Magnitude of the wind velocity.

Wind Direction: Direction from which the wind blows as measured in degrees clockwise from true north.

Day Type: This is the type of day you want to use when performing a simulation with steady airflows and no weather file. The Day Type corresponds to one of the twelve day schedules that you can create.

Weather and Wind – Location Properties

Longitude, latitude, and time zone are currently not used by CONTAM. They have been included for potential use in the future if heat transfer analysis is implemented within the program.

Altitude: ContamW uses the site altitude to determine a default absolute barometric pressure for the steady state weather data.

Weather and Wind – Wind Pressure Display Properties

The wind pressure display values are used to calculate the wind pressures displayed on the screen in wind pressure display mode. You activate the wind pressure display mode by selecting **Wind Pressure** from the **View** menu.

NOTE: These values are not used by ContamX when performing a simulation. To display the wind pressure results use the **Wind Pressure** selection of the **View** menu. This feature is provided so that you can verify the wind speed and direction information visually on the SketchPad. (see Checking Wind Pressure Data in the Defining Steady State Weather and Wind section)

Ambient Temperature: The ambient temperature is used to calculate outside air density and building stack effects.

Absolute Pressure: Be sure to enter the absolute barometric pressure instead of pressure corrected to sea level.

Wind Speed/Direction: Enter the wind speed for the site. The wind direction is calculated in degrees clockwise from true north.

Day Type: This is the day type as defined in the Week Schedule .

3.16 WORKING WITH WPC FILES

Wind Pressure and Contaminant (WPC) files can be used to account for the variation of external wind pressures and outdoor contaminant concentrations over the building envelope. This method addresses the need to allow for the use of general, spatially varying wind pressure and ambient contaminant concentrations such as those from wind tunnel experiments or atmospheric models, e.g., plume or puff dispersion simulation tools. The WPC file provides exterior pressure and/or contaminant concentrations time histories for every flow path that connects to the ambient zone including duct terminals and outdoor air intakes of simple air handling systems.

The WPC files are created externally to CONTAM, however their creation can be assisted by ContamW that can generate a Path Location Data (PLD) file listing the location of all the airflow paths that are connected to the ambient zone. The following figure illustrates the interaction between CONTAM and the WPC file. The dashed lines in the figure represent optional components. The WPC file is an ASCII file that could be created using conventional means, e.g., spreadsheet converted to text. One could also develop a WPC File Converter program to create the files from External Wind Pressure and Contaminant data files created by a separate tool, e.g., exterior CFD program. A converter could then work with the PLD file to create a WPC file specific to the building in question, i.e., the PRJ file. ContamW provides a means to activate a user-selectable converter. The details of the WPC and PLD file formats are presented in the sections titled WPC File Format, and PLD File Format respectively. CONTAM does not include either an EWC file creation tool or an EWC-to-WPC converter tool.

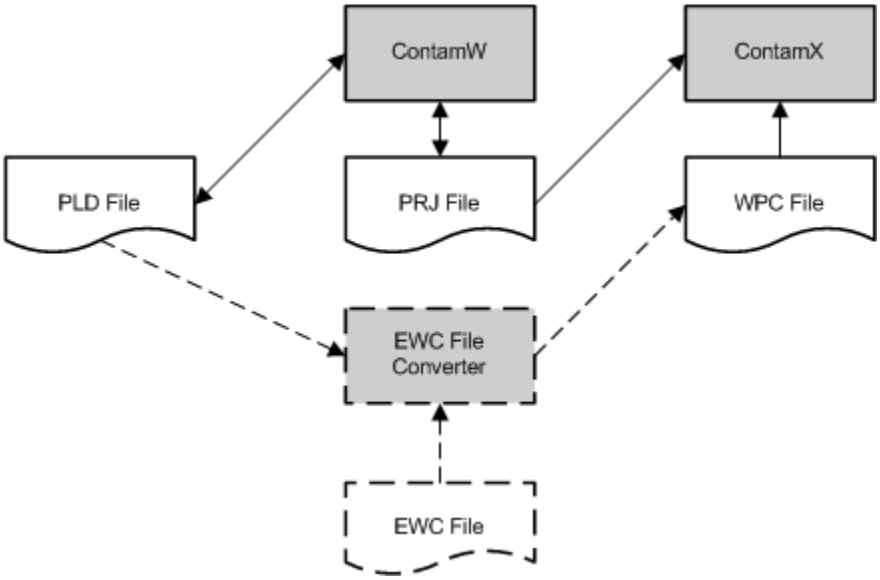


Figure - WPC File Implementation Schematic

Steps to implement WPC files:

1. Enter Location coordinates for each penetration of the building envelope, i.e., connections between indoor and outdoor zones, including: airflow paths, duct terminals and outdoor air intakes of simple air handling sytems.

2. Create a WPC File according to the WPC File Format provided later in this section using either an external editor or a user-developed *EWC File Converter* program.

   If using an external editor, you can use the Create WPC File dialog box to generate a list of path location data that contains the locations of all the building envelope openings. You can also have ContamW call your EWC File Converter program via the Create WPC File dialog box that is presented later in this section and activated via the **Weather→Create WPC File...** menu command.

3. Set the simulation to utilize the WPC file (see Simultion Weather Properties in the Working with Simulations section).

4. Verify WPC file

   If you have created the WPC file using an external editor, you should verify that its contents match that of the project file. You can do this using the **View → Wind Pressure Mode** to display the WPC wind pressure type (see Checking Wind in the Defining Weather and Wind section) or initiating the pre-simulation checks by selecting either the **Run Simulation** or **Create ContamX Input File** from the **Simulation** menu.Either option will compare the WPC and PLD files and provide notification of discrepancies between them including location data and contaminant data that do not match. If you use and EWC file converter program, then the verification step will occur automatically. If there are any flow paths that do not map to a location in the WPC file, they will be highlighted on the SketchPad and a list of these locations will be written to the CONTAMW2.LOG file. Look for list of Paths/Terminals/Junctions not found in WPC file.

5. Run simulation

   If there are errors between the WPC file and the PLD file, then ContamX will not be able to run a simulation. ContamW will not initiate ContamX. If ContamX is run from the command line on a project having discrepancies, it will also exit with an error message.

3.16.1 Create WPC File Dialog

The Create WPC File dialog box is a tool that can aid in the creation of WPC and associated files.You can use this dialog box whether you are creating WPC text files using an external editor or using an *EWC File Converter*. Access this dialog box via the **Weather → Create WPC File...** menu item.

NOTE: WPC files can be used during a simulation in conjunction with the transient weather and transient contaminant files as follows:

If using wind pressures from a WPC file, you may not use a WTH file.

If using contaminant concentrations from a WPC file, you may not use a CTM file.

Name of WPC File to create: This is the name of the WPC file to create using the *EWC File Converter*. You can either use the *Browse...* button to select an existing file, or enter a new name.

WPC Description: Enter a description you want an *EWC File Converter* to write when creating a new WPC file.

EWC File Name: The ***E****xternal* ***W****ind Pressure and* ***C****ontaminant* (EWC) file contains the external contaminant concentrations and pressures that the *EWC File Converter* (which must be

developed for your specific application) will convert into a WPC file. Use the *Browse...* button to select an existing EWC file.

WPC File Creation Program: Use the *Browse...* button to select a program to use to create a WPC File for example an *EWC File Converter*.

Coordinate Transformation Data

These parameters can be used by an *EWC File Converter* to establish the relationship between the coordinates of the EWC file and the CONTAM project file. No transformation is required if the coordinate systems for the PRJ and EWC files are consistent. These values are stored in the PLD file by ContamW.

Origin (X, Y and Z): The location of the origin of the CONTAM PRJ file with respect to the origin of the EWC file.

Rotation Angle: The rotation of the x and y axes of the EWC coordinate system about the z axis to align with the x and y axes of the CONTAM coordinate system. Counter-clockwise is considered the positive direction.

Conversion Tolerance

These are the tolerances that an *EWC File Converter* might use to determine how closely information in the EWC file and the PLD file must match in order to resolve contaminant (species) and location data. These values are stored in the PLD file by ContamW.

Species: Species can be resolved by their molar mass, i.e., the molar mass of each species defined in ContamW and the WPC file must not differ by more than this amount to be considered the same species.

Location: Locations can be resolved by their distance as determined by an *EWC File Converter*, e.g., distance to center of a grid cell. Units within the PLD file are in meters.

Date and Time

These parameters can be used by an *EWC File Converter* to determine the date and time values written to the WPC file for the convenience of working according to CONTAM's schedules. These values are stored in the PLD file by ContamW.

Data Time Shift: The format for this value is hh:mm:ss. An *EWC File Converter* could add this value to the EWC file time. For example an *EWC File Converter* could create a file that begins at 00:00:00 and set the next time to be the *Data Time Shift* you enter here. The data for these two times would be the initial values in the EWC file.

Start and End Date: The format for this value is mmmdd. For example, enter January 1 as Jan01. An *EWC File Converter* could use this to output the date(s) to use when running transient simulations.

Include in WPC File

Wind Pressures: Check this box to have a converter create a WPC file that contains wind pressure data.

Contaminant Concentrations: Check this box to have the converter create a WPC file that contains contaminant data. During a simulation contaminant concentrations in the WPC file take precedence over other transient contaminant settings. For those contaminants that are

defined within the PRJ file but not included in the WPC file, the steady-state contaminant data will be used, i.e., the default concentrations.

Generate WPC File: Use this button to activate the user-developed WPC Creation Program. This will first perform checks on the PRJ file and generate a PLD file if all envelope openings are properly defined.

Generate PLD File: Use this button to manually generate a PLD file. The PLD file will be stored in the same location as the PRJ file and have the same name as the PRJ file but a PLD extension. This file is useful for creating a WPC file with an external editor.

3.16.2 Envelope Opening Locations

When using WPC Files you must enter the coordinates for each opening connected to the ambient including: airflow paths, duct terminals, and outdoor air intakes of simple air handling systems. Coordinate data is input via the Wind Pressure Property Pages of these building components. The location information will be used to create the PLD File which in turn is used to verify that there is matching information within the WPC File prior to performing a simulation.

NOTE: Even though every opening connected to ambient requires location data when using WPC Files, they are not required to be unique. They simply must match a location within the WPC File.

Airflow Paths and Duct Terminals

Enter the X and Y coordinates and units for each airflow path and duct terminal. The Z coordinate will be calculated by ContamW to be the Relative Elevation plus the Elevation of the building level on which the path or terminal is located.

Outdoor Air Intakes of Simple Air Handling Systems

Enter the X, Y and Z coordinates and units for the outdoor air intake of each Simple AHS. Enter the Z coordinate as the height of the midpoint of the outdoor air intake with respect to the building reference height.

3.16.3 WPC File Format

This file provides ambient pressure and contaminant concentrations values for every flow path that connects to ambient.

There are two versions of the WPC file 2.1 and 2.4. For version 2.1, pressures must be input as absolute pressures in units of Pa. Wind pressure will be computed from the ambient pressure and density provided at each time interval in the WPC file and the absolute pressure and elevation of each flow path as follows:

$$P_{wind} = P_{path} - P_{ambt} + \rho \bullet g \bullet Z_{path}$$

For 2.4 version files the pressure in the WPC file will be the wind pressure itself in units of Pa.

Contaminant concentrations must be provided in units of kg of contaminant per kg of air. See Concentration Conversions in the Contaminant Files subsection of the Working with Species and Contaminants section for conversion factors.

The first line of the WPC file is used to identify the type of file. For a 2.1 format WPC file it is exactly:

```
WPCFile ContamW 2.1
```

Or for a 2.4 format WPC file it is exactly:

```
WPCFile ContamW 2.4
```

The second line is a description of the file entered by the user:

```
desc[] // file description (I1) {W}; may be blank
```

Succeeding lines contain:

```
nPath // number of flow paths (I2)
nCtm // number of contaminants (species) (I2) [0 possible]
UsePres // 1 = use pressures, 0 = don't (I2)
dtmin // minimum time step (I2)(not used by ContamX)
StartDate // first date for WPC data (mm/dd → IX)
EndDate // last date for WPC data (mm/dd → IX)
I1 Name[1] // name of contaminant 1 (I1 - 16 characters max)
I1 Name[2] // name of contaminant 2 (I1)
...
Name[nCtm] // name of contaminant NumCont (I1)
```

The next NumPath lines describe each flow path:

```
     number // sequence number (I2)
     X // X-coordinate (R4)
     Y // X-coordinate (R4)
     Z // Z-coordinate (R4)
     map // mapping [0 = OK; 1 = error] (I2)
```

For each day StartDate to EndDate:

For each time 00:00:00 to 24:00:00 of each date:

First line:

```
  date // date (mm/dd → IX)
  time // time of day (hh:mm:ss → I4)
  Pamb // ambient pressure [Pa] (R4) (if pressures are used)
  dens // air density [kg/m3] (R4) (if pressures are used)
```

Second line: (if pressures are used)

```
  P[1] // pressure at path[1] [Pa] (R4)
  ...
  P[nPath] // pressure at path[nPath] [Pa](R4)
```

Third line (for first contaminant, if nCtm > 1):

```
  C[1] // concentration at path[1] [kg/kg air] (R4)
  ...
  C[nPath] // concentration at path[nPath] [kg/kg air] (R4)
```

Fourth line (for second contaminant, if needed):

```
  ...
```

Last line (for contaminant nCtm, if needed):

```
  C[1] // concentration at path[1] [kg/kg air] (R4)
  ...
  C[nPath] // concentration at path[nPath] [kg/kg air] (R4)
```

The file description may not begin with a "!". The StartDate and EndDate are used to verify that the file data covers the entire period to be simulated. The StartDate may not be later than the EndDate.

The data must start at time 00:00:00 on the StartDate and end at 24:00:00 on the EndDate. The times must be in consecutive order, but the difference between successive times need not be constant.

Example:

```
WPCfile ContamW 2.1
For WPCtest3.prj
2 ! flowpaths
1 ! contaminants
1 ! use pressure flag
0 ! time step
01/01 ! start date
01/01 ! end date
C1
!nr X Y Z map
1 0.000 4.000 1.500 0
2 8.000 4.000 1.500 0
01/01 00:00:00 101325 1.204
101308.30 101306.30
0.0 1.0e-6
01/01 24:00:00 101325 1.2041
101308.30 101306.30
0.0 1.5e-6
```

3.16.4 PLD File Format

The PLD (Path Location Data) File will consist of the locations of any flow paths that connect the building to the external environment, as well as additional information for directing the behavior of the EWC File Converter. This file is created by ContamW and stored in the same directory as the project file having the same name as the project file but with the .pld extension replacing the .prj extension. It is created as needed for use in comparing WPC file information with that in the CONTAM project file. (See Working with WPC Files)

The file is in an ASCII line-delimited format.

The reference location, defined by (Xref,Yref,Zref), represents the EWC file location that is equivalent to the origin of the CONTAM coordinate system for the path locations. The user must coordinate between the EWC file and PRJ file to determine this location's coordinate values.

The rotation angle is about the Z (vertical) axis. The user must coordinate the direction of rotation between EWC and PRJ file coordinates It is assumed there are no rotations about the X or Y axes.

The six coordinates on line 6 define a bounding box that surrounds the flow paths within this file, which could be used to assist the EWC File Converter in rapidly excluding unneeded points (e.g.,

that are too distant from the building) during the mapping process. Note that this bounding box does not necessarily surround the entire building.

The simulation time step provides a recommendation to the EWC File Converter for reducing the size of the WPC file, so that only the time steps that coincide with the simulation time step could be included (which could reduce interpolation required).

Each flow path is uniquely identified by a combination of flow path type and flow path ID values.

Comments begin with an exclamation point ('!') and may begin at the start of any line (so that the entire line will be ignored) or after all fields of a line (so that the remainder of the line will be ignored).

For user readability, fields should be commented whenever possible.

The format of R4 data types may be interpreted similar to a C scanf() statement's conversion specification.

Header Section:

First three lines:

```
EWC filename   // (I1 full path)
WPC filename   // (I1 full path)
desc[]  // file description (I1)
```

The next header line is exactly:

```
! Xref    Xref    Yref    angle
```

The next line contains reference coordinates in meters:

```
Xref   // (R4 %7.3f)
Yref   // (R4 %7.3f)
Zref   // (R4 %7.3f)
rotation angle  // Real format %8.3f
```

The next line contains latitude and longitude in units of degrees:

```
latitude   // (R4 %8.4f) positive for North
longitude  // (R4 %8.4f) positive for East
```

The next bounding box header line is exactly:

```
! Xmin    Xmax    Ymin    Ymax     Zmin    Zmax
```

The next line contains bounding box coordinates:

```
Xmin   // (R4 %7.3f)
Xmax   // (R4 %7.3f)
Ymin   // (R4 %7.3f)
Ymax   // (R4 %7.3f)
Zmin   // (R4 %7.3f)
Zmax   // (R4 %7.3f)
```

The next time step heading line is exactly:

```
! step  shift     start   end
```

The next line contains time step data:

```
time step  // (I4) [s]
data shift // (hh:mm:ss → I4) start time for EWC when converted to WPC
start date // (mm/dd → IX)
end date   // (mm/dd → IX)
```

The next line contains:

```
wind press flag    // (I4) 1: include pressures in WPC, 0: don't
number of species  // (I4)
species map tolerance // (R4 %7.3f) molar mass tolerance
```

Contaminant Definition Section:

The Contaminant Definition Section begins with a heading line of exactly:

```
! name  m.wt
```

The next (number of species) lines contain for each species:

```
name[]    // (I1 - 16 characters max)
molarmass // (R4 %6.2f)
```

Airflow Path Definition Section:

The first line of the Airflow Path Definition Section contains:

```
number of flow paths // (I4)
path map tolerance   // (R4 %7.3f) [m]
```

The next heading line is exactly:

```
! id#   X         Y       Z
```

The next (number of flow paths) lines contain for each airflow path:

```
path number    // (IX)
x-coordinate   // (R4) [m]
y-coordinate   // (R4) [m]
z-coordinate   // (R4) [m]
```

The last line of the file is exactly:

```
-999
```

Example:

```
C:\Program Files\Contamw2\Prjs\WPCcube.ewc ! EWC file
C:\Program Files\Contamw2\Prjs\WPCcube1.wpc ! WPC file
WPC description
! Xref    Xref    Yref    angle
  0.000   0.000   0.000   0.00
  0.0000  0.0000  ! no latitude/longitude
! Xmin    Xmax    Ymin    Ymax     Zmin    Zmax
  0.000   5.000   2.500   10.000   1.000   1.000
! step  shift     start   end
  0     00:00:00  1/1     1/1
  1     2         0.01 ! pressures flag, # species, mapping tolerance
! name  m.wt
  CO    28.00
  CO2   44.00
  4     1.00 ! number of flow paths and mapping tolerance
! id#   X         Y       Z
  1     5.000     10.000  1.000
  2     0.000     7.500   1.000
  3     0.000     5.000   1.000
  4     0.000     2.500   1.000
-999
```

3.17 WORKING WITH SCHEDULES

Schedules provide you with the ability to modify various parameters based on the time of day and day of the week. You use schedules to set or modify values of controlled parameters at times which you define. You can schedule parameters such as the flow through airflow paths; ducts; simple air handling system inlets, outlets and fraction of outdoor air intake; contaminant source/sinks; the location and contaminant generation rate of occupants; and zone temperatures.

CONTAM provides occupant schedules and non-occupant schedules which are split up into week schedules which are in turn made up of one or more day schedules. Non-occupant week schedules can be associated with as many schedulable items as you need, whereas occupant week schedules can only be applied to a single occupant. Only non-occupant schedules can be shared between CONTAM project files using the ContamW Library Manager. Occupant schedules can not be shared between projects, because they are dependent on the level and zone names which may be different between projects. The following sections provide detailed information related to the creation, modification and deletion of schedules in ContamW.

Schedules apply when performing both transient and steady-state simulations, and schedule values are accessed according to the Day Type and time during a simulation. The Day Type is used to select one of twelve day schedules that make up each week schedule as explained below. The Day Type will be determined during simulation via either the steady state or transient weather data (See Run Control Properties in the Working with Simulations section). For greater scheduling flexibility, use the CVF and DVF control types which allow scheduling to vary over the course of a year (see CVF/DVF in the Working with Controls section).

If a schedule and a control signal (see Working with Controls) are both associated with the same building element icon, then the control signal will take precedence over the schedule during simulation.

3.17.1 Creating Schedules

You can create schedules using either the CONTAMW Library Manager (see Working with Data and Libraries) or wherever a "New Schedule" button is provided when working with various building component property sheets (e.g. airflow paths, ducts, source/sinks, and occupant icons).

Non-occupancy Schedules

You create non-occupant schedules by first selecting to create a new week schedule to display the Week Schedule dialog box. You must then define one or more day schedules that you then associate with each day of the week schedule. Each week schedule consists of 12 days – one for each day of the week and five more days that you can use for special situations such as holidays. These are refered to as the 12 day types. New day schedules are created from the Week Schedule dialog box by clicking on the "New Schedule" button. This will display the Day Schedule dialog box whose features are explained below.

Temperature Schedules

Temperature schedules are a type of non-occupant schedule. Unlike other non-occupant schedules they have units associated with them. Temperature schedules are only associated with zones and duct junctions and are considered separately from other non-occupant schedules when creating and editing. For instance, when displaying a list of temperature schedules, only those

non-occupancy schedules having temperature units will be displayed (unitless schedules will not be displayed).

Occupancy Schedules

You create occupant schedules by first creating an occupant exposure icon (see Creating Occupants). You then click on the "Edit Occupancy Schedule" button that is provided on the Occupant Data property page of the "Occupant Exposure Properties" property sheet.

3.17.2 Modifying Schedules

You modify existing non-occupancy week and day schedules through the Week Schedule and Day Schedule dialog boxes respectively. Edit occupancy schedules by editing the properties of existing occupant exposure icons. The details of editing schedules are presented in the following sections.

3.17.3 Deleting Schedules

You can only delete occupancy week schedules by deleting the associated occupant exposure icon. Deleting all other schedules can only be accomplished using the CONTAMW Library Manager (see Working with Data and Libraries) which will only allow you to delete those schedules that are no longer associated with any occupants, zones or other building components.

3.17.4 Week Schedule Properties

These are the properties and methods you use to define non-occupancy week schedules. You must associate a day schedule with each day of a week schedule (7 days of the week plus 5 extra days). The extra days are provided for any day that may have a non typical schedule, such as a holiday. Once defined, you may display the schedule graphically by pressing the "Display Graphically" button.

Name: This is the name you give to this week schedule. This name must be unique among the week schedules within this project.

Units: Select the temperature units to use when defining a zone temperature schedule. This control will only be available if these properties are accessed from a zone property sheet, Zone Data or via the **Data → Week Schedules ▸ Temperature…** menu selection.

Description: Field for entering a more detailed description of the specific schedule.

Day Schedules: Every "Week Schedule" consists of twelve days (day type 1 - 12) that must be associated with "Day Schedules". These day schedules may or may not be unique for each of the 12 days. If the project has no available day schedules a new day schedule must be defined. Do this by pressing the "New Day Schedule" button, then editing the day schedule as explained in the section entitled "Day Schedule." You can edit existing day schedules by selecting them in the "Available Schedules" list box and then pressing the "Edit Day Schedule" button, then following the procedure explained in the Day Schedule section. If a day has already been associated with a day schedule you can change it by first highlighting the day to be changed in the Day Schedules list box, highlighting the day schedule to replace the existing schedule, and then pressing the "<< Replace <<" button.

Only those day schedules of the same type as the week schedule will appear in the list of day schedules, i.e., dimensionless or temperature.

3.17.5 Day Schedule Properties

These are the properties and methods you use to define day schedules. You may display the schedule graphically by pressing the "Display Graphically" button.

Name: This is the name you give to this day schedule. This name must be unique among the day schedules within a project.

Description: Field for entering a more detailed description of the specific schedule.

Shape: Use this check box to set how CONTAMW interprets values between the data points you define.

- Rectangular: characterized by a step function between the data points
- Trapezoidal: characterized by linear interpolation between the data points.

Schedule Data: These are the time/multiplier pairs that you set to define a day schedule. The times are entered in the hh:mm:ss format.

For unitless schedules, the value is a multiplier between 0.0 and 1.0 used to adjust down the maximum design values of various schedulable building components, e.g., constant volume flow rate of a fan flow element.

To insert a new data point
Enter the new time/multiplier pair in the "Time" and "Value" edit boxes in the Edit Schedule Value Section of the dialog box and press the "<< Insert <<" button. The data point will be inserted into the schedule in the correct sequence.

To modify an existing data point
Select the data point that you wish to modify from the list, enter the revised data, and press the "<< Insert <<" button to overwrite the previously entered data point.

To delete an existing data point
Select the data point you want to delete from the list and click the "Delete" button. You can not delete 00:00:00 or 24:00:00.

3.17.6 Occupant Week Schedule Properties

These are the properties and methods you use to define occupant week schedules.

Description: Field for entering a more detailed description of the specific schedule.

Day Schedules: Every "Occupancy Week Schedule" consists of twelve days (12 day types) that must be associated with Occupant Day Schedules. These schedules may or may not be unique for each of the twelve day types. If the project has no available day schedules a new day schedule must be defined. This is done by pressing the "New Day Schedule" button then editing the day schedule as explained in the Occupant Day Schedule section. You may edit existing day schedules by selecting them in the "Available Schedules" list box, pressing the "Edit Schedule" button, and then following the procedure explained in the Occupant Day Schedule section. If a day has already been associated with a day schedule, you may changed it by first highlighting the day to be changed in the Day Schedules list box, then highlighting the day schedule to replace the existing schedule, and finally pressing the "<< Replace <<" button.

3.17.7 Occupant Day Schedule Properties

These are the properties and methods you use to define occupant day schedules.

Name: This is the name you give to this occupant day schedule. This name must be unique among the occupant day schedules within this project.

Description: Field for entering a more detailed description of the specific schedule.

Schedule Data: These are the time/location data points that you set to define an occupancy day schedule. The times are entered in the hh:mm:ss format, and you must provide leading and trailing zeros; for example 9:30 am would be entered as 09:30:00 not as 9:30:00. The location consists of a level and zone in which the occupant resides for the indicated time period. Select a value of "null" from the zone list to indicate the occupant is not in the building for a given time period.

Location data will also include zone coordinates if the occupant enters a 1D convection diffusion zone. For such cases, enter the X, Y and Relative Elevation (Z) coordinates and units for each 1D zone the occupant enters. X and Y are absolute coordinates and Rel Elevation is relative to the level of each1D zone that the occupant enters.

To insert a new data point
Enter the new time and location data in the "Modify Data" block, press the "<< Insert <<" button and the data point will be inserted into the schedule in the correct sequence.

To modify an existing data point
Select the data point that you wish to modify from the list, enter the revised data, and press the "<< Insert <<" button to overwrite the previously entered data point.

To delete an existing data point
Select the data point you want to delete from the list and click the "Delete" button. You can not delete 00:00:00 or 24:00:00.

3.18 WORKING WITH SIMULATIONS

CONTAM allows you to perform several types of simulations. The type of simulation you wish to perform depends on the goal of your analysis. You can perform simulations to determine just inter-zone airflow rates and pressure differences, or you can also have CONTAM solve for contaminant concentrations within each zone of a building. CONTAM provides the ability to perform several combinations of steady state, transient and cyclical simulations of both airflow and contaminant dispersal.

The following table shows the different combination of simulation methods you can apply using CONTAM.

Airflow Simulation	Contaminant Simulation	Purpose
Steady State	None	Used to obtain airflow rates and pressure differentials under constant building system and/or weather conditions.
Steady State	Steady State	Used to obtain equilibrium contaminant concentrations under conditions of steady airflow.
Steady State	Transient	Used to obtain a time history of contaminant concentrations under conditions of steady airflow. This is similar to the previous table entry but will allow you to observe the contaminant concentrations for each time step leading up to the final time step.
Steady State	Cyclic	Used when it's important for initial contaminant concentrations to match the final concentrations under steady airflow conditions. This provides a method of simulating a typical pattern of operation by simulating only a representative time period (e.g. one day).
Transient	None	Used to obtain a time history of airflow rates and pressure differentials under changing building system and/or weather conditions.
Transient	Steady State	Not allowed
Transient	Transient	Used to obtain time histories of airflow rates, pressure differentials and contaminant concentrations under changing building system and/or weather conditions.
Transient	Cyclic	Used when it's important for initial contaminant concentrations to match the final concentrations under transient airflow conditions. This provides a method of simulating a typical pattern of operation by simulating only a representative time period (e.g., one day).
Steady State OR Transient	Short Time Step Method (Transient OR Cyclic)	The *short time step* contaminant solution method is available beginning with version 2.4. This is an explicit solution method useful when interested in rapidly changing conditions that require short time steps to capture. It is required when using the 1D zone and duct models.
Steady State OR Transient	CVODE (Transient only)	This variable time step ordinary differential equation solver may provide a more accurate solution, especially in cases with stiff contaminant transport systems. Stiff systems are those with wide variations in system dynamics, e.g., air change rates or kinetic reactions.
Duct Balance	n/a	Use this feature to have ContamX adjust duct terminal balancing coefficients to provide user-defined design airflow rates at each balancing terminal. This does not perform a simulation per-say; it balances the system for upcoming simulations.
Building Airflow Test	n/a	Generates a set of data, mostly related to building ventilation, that enables you to review the reasonableness of your building model and the input values.
Pressurization Test	n/a	Performs an automated, idealized fan pressurization test to determine building envelope tightness of your building model.

To perform a simulation, you *set the simulation parameters*, *run the simulation*, and *view simulation results*. Simulation parameters are categorized by *run control*, *numerics*, and *output*

properties. *Run control* properties include the simulation methods, simulation dates and times, type of weather to use (steady or transient), and simulation time step. *Output properties* are used to set the type and amount of data you wish to have sent to the simulation results files. *Numerics properties* include convergence criteria, maximum iterations and relaxation coefficients for both airflow and contaminant equation solvers. ContamW provides default settings for these parameters. However, you will probably work more with the r*un control* and *output parameters* than the *numerics parameters*.

Setting Simulation Parameters

To set simulation parameters you use the **Set Simulation Parameters…** selection of the **Simulation** menu or the **F4** keyboard shortcut. This will display the "Simulation Parameters" property sheet. Detailed descriptions of all simulation properties are given in the Simulation Parameters section of this documentation.

Running Simulations via ContamW

Once you have set the simulation parameters, you can run the simulation. To run a simulation select **Run Simulation** from the **Simulation** menu. This actually launches a separate program, ContamX, which in turn displays the "CONTAMX Simulation Control" dialog box.

While there are many factors that can affect the ability of ContamX to quickly and successfully obtain a solution (*converge*), there are situations that can cause the simulation to not converge or to converge very slowly. The relative magnitudes of the flow coefficients have some effect on the convergence of the nonlinear equation solver of ContamX [Wray and Yuill 1993]. Airflow networks containing paths whose resistances differ by many orders of magnitude will typically require more iterations than networks with paths of similar resistances. Reducing the relaxation coefficient from its default value of 0.75 to a value nearer one-half can often help convergence of these more difficult cases. The maximum number of iterations can also be increased. However, the most effective way to improve convergence is to remove those very low resistance paths that are not likely to effect the computed airflow rates. For example, when simulating a shaft (see Shafts), represent a shaft using phantom zones as opposed to using airflow elements with large cross-sectional areas between zones on adjacent levels.

Running Simulations via Command Line

CONTAM utilizes the ContamX stand-alone simulation engine, contamx3.exe. ContamX utilizes the same .PRJ file that you create using ContamW. With the ContamX program, you can perform batch mode operations to process multiple project files. Or you can create your own file processor to modify project files, for example to perform sensitivity analysis by performing multiple runs while varying desired building parameters. If you wish to modify the .PRJ file in such a manner, you should become familiar with the .PRJ file format that is now annotated with comment lines to improve the legibility of the file. (see Appendix A - PRJ File Format)

To obtain a PRJ file for command-line use, you can simply create and save the project in the usual manner using ContamW. To check that the project is “well formed”, you can either use ContamW to run a simulation with the file, or use the **Run Building Check** selection of the **Simulation** menu. This is especially important if new controls have been added to the project, because ContamW must determine the proper evaluation sequence of the controls. Sequence values will be saved to the project file for each control.

Building Check

Note this building check is also performed prior to initiating a simulation from within ContamW via the **Simulation → Run Simulation** menu command. If during the *building check* routine ContamW detects problems with your project, it will provide you with feedback to assist you in correcting the problems. This feedback will be provided by displaying the **Building Check** dialog box which will display a list problems found and allow you to highlight problematic icons on the SketchPad as necessary.

Viewing Simulation Results

Once you have successfully run a simulation, you can use ContamW to view simulation results or to create files that can be imported into a spreadsheet. These capabilities are explained in the Working with Results section.

3.18.1 CONTAMX Simulation Control

The "ContamX Simulation Control" dialog box allows you to control and view the status of a simulation. Once you have set the simulation parameters, activate the CONTAMX Simulation Control dialog via the **Simulation → Run Simulation** command from the ContamW menu. Press the "Start Simulation" button to run the simulation. If at any point during the simulation you want to terminate the simulation, press the "Stop Simulation" button.

Simulation Status: The current status of the simulation is displayed in this section. The date, time, day, and the current iteration of the simulation are shown. A check box is provided to terminate cyclic simulations at the end of the next cycle/day. Use this check box if it appears the simulation is not converging, or you feel enough iterations have been performed.

Simulation Settings: This field displays a summary of the current Run Control simulation parameters. This is provided for convenience to enable you to review the type of simulation that CONTAM is currently configured to perform.

Start Simulation: Press this button to start a simulation. Once you start a simulation, any existing results files for the current project file will be deleted. If you wish to save the current result files, you should either rename the files, move them to a location other than the directory in which the project file exists, or rename the current project file so that new results files are created associated with the new name.

Stop Simulation: Press this button to immediately terminate a simulation.

3.18.2 Simulation Parameters

Simulation parameters are those used to control the type of simulation that you wish ContamX to perform and the type of results output you wish ContamX to provide. This section provides detailed descriptions of the specific simulation parameters.

Simulation – Run Control Properties

Simulation Method

Set the type of simulation you wish to perform for both Airflows and Contaminants by checking the appropriate radio buttons. Refer to the Working with Simulations section for a table of allowable combinations of simulation method settings.

Airflows: You can use one of the following airflow calculation methods.

- **Steady:** A steady state simulation will utilize the weather set by pressing the Edit Weather Button on the Run Control property page. This will calculate a single set of airflows. If any building components have schedules associated with them, then schedule values will be determined by the *Day Type* set in the *Steady State Weather Data* (see Weather Properties under Working with Weather and Wind) and the *Steady Simulation Time*.
- **Transient:** Transient simulations perform calculations over a user-defined period of time at a regular time interval referred to as the calculation time step. When performing a transient simulation CONTAM can use either steady state weather data or weather and contaminant files that define the weather and ambient contaminant concentrations for the time period of the simulation. See Defining Transient Weather and Wind in the Working with Weather and Wind section for more details.
- **Duct Balance:** Use this feature when working with detailed duct systems to have CONTAM adjust terminal duct balancing coefficients to provide user-defined airflow rates at the duct terminals. See Working with Ducts.
- **Building Airflow Test:** The *building airflow test* method performs calculations that aid in verifying data input to your building model. Use this test prior to performing analysis with your building model to verify system and total airflow rates to each zone, zone temperatures and whole building air change rate. You must have only one contaminant set to be used in the simulation, and it must be a trace contaminant, when you run this test. Results of the test will be written to the validation file having the same name as the project file and the .val extension (see Working with Results). This file will be overwritten each time either the *Building Airflow Test* or *Building Pressurization Test* is performed.
- **Building Pressurization Test:** The building pressurization method performs an automated whole-building fan pressurization test (see Special Applications of CONTAM) to determine building envelope airtightness under idealized test conditions. See the Special Applications of CONTAM section for a detailed discussion of building pressurization tests.
  ContamX will temporarily set conditions for the duration of the building pressurization calculation as follows:
  - no-wind
  - ambient pressure set to steady state barometric pressure
  - ambient and interior zones set to 20 ºC
  - conditioned zones (included in building volume) set to constant test pressure
    NOTE: This will eliminate the effect of airflow between building zones as explained in the Special Applications of CONTAM section.
  - unconditioned zones (not included in building volume) set to variable pressure
  - “close” all duct terminals and duct leaks
  - “turn off” all forced flow elements

    When you perform a simulation using this method, ContamX will perform a steady state airflow calculation under the above conditions and calculate the total airflow rate between conditioned and unconditioned zones of the project. Results of the test

will be written to the validation file having the same name as the project file and the .val extension (see Working with Results). This file will be overwritten each time either the *Building Airflow Test* or *Building Pressurization Test* is performed.

Contaminants: You can choose one of the following contaminant simulation options:

- **None:** Do not perform contaminant analysis, only airflow analysis.
- **Steady:** The steady contaminant option can only be used in conjunction with steady airflows and will run until the system reaches equilibrium. Contaminant sources are modeled as continuous sources with a constant generation rate. If any sources have schedules associated with them, then schedule values will be determined by the *Day Type* set in the *Steady State Weather Data* (see Weather Properties under Working with Weather and Wind) and the *Steady Simulation Time*. Burst sources will be ignored. Ambient contaminant concentrations will be the *Default Concentration* values of each contaminant.
- **Transient:** Transient simulations performs calculations over a user-defined period of time and at a regular time interval referred to as the *calculation time step*. When performing a transient simulation CONTAM can use either constant ambient concentrations as given by default contaminant concentrations or dynamic concentrations from external contaminant files. See Contaminant Files and Working with WPC Files.
- **Cyclic:** A cyclic simulation repeats a 24h cycle until steady-periodic conditions are achieved. As with transient simulations, cyclic simulations can implement either steady state or dynamic weather and contaminant data. If using dynamic weather or contaminant data files (CTM and WTH), data will be obtained from the file for the date indicated by *Steady & Cyclic Simulations* date.

Transient Integration Method: Select the type of transient or cyclic simulation to perform from the available options. More detailed options are available via the Contaminant Numerics simulation parameters.

- **Default Solver (Implicit Euler):** This is the traditional, fixed time step solver.
- **Short Time Step (Explicit Euler):** You can use this method with either the Transient or Cyclic contaminant simulation methods. This method is required to perform simulations of one-dimensional convection diffusion zones and/or ducts. See Contaminant Numerics Properties for specific control in applying the *short time step* method to 1D ducts and 1D zones.
  NOTE: This is an explicit solution method that should only be used for short time steps, therefore there are some checks performed to help avoid known stability problems. ContamX will check the zone airflow rates (of all zones having non-zero volumes) during airflow simulation to ensure that they do not exceed one air change per time step. If this limitation is exceeded for any zone, then the simulation will be terminated with an error message indicating the offending zone(s). The problem could also manifest itself with an error message indicating an invalid zone density was calculated due to the calculation of a negative absolute pressure.
- **CVODE:** This is a variable time step ordinary differential equation solver [Hindmarsh and Serban 2012]. It may provide a more accurate solution, especially in cases with stiff contaminant transport systems. Stiff systems are those with wide variations in system

dynamics, e.g., air change rates or kinetic reactions [Lorenzetti 2013].
NOTE: This solver is currently limited in that it will not handle projects that contain 1D zones, CFD zones, occupant exposure, burst sources, or zones (nodes) with zero volume. Further, the CVODE solver does not support the bridge mode (socket communication capabilities).

Reset initial contaminant concentrations to…: Use this feature to reset the initial contaminant concentrations of all zones, including the implicit supply and return zones of simple air handling systems, prior to running a simulation. You can reset initial concentrations to *Zero*, *Ambient* or *Current Results Concentrations*. If you do not check one of these boxes, ContamX uses the Initial Concentrations indicated on the Contaminant Property page for each zone (See Contaminant Data under the Working with Zones section). Checking one of these boxes causes the Initial Concentration values for each zone to be overwritten.

- **Zero:** This option sets the initial contaminant concentrations to zero.
- **Ambient Concentrations:** This option sets the initial contaminant concentrations to the default contaminant concentrations as set when defining Project Species (See Species Properties).
- **Current Results Concentrations:** This option sets the initial contaminant concentrations to be the same as the concentrations in each zone at the time step for which ContamW is currently displaying results on the SketchPad. You must be in the results viewing mode to utilize this feature. The current time step is displayed in the middle pane of the Status Bar (See the Status Bar section). This option is especially useful for cyclic simulation, where you may continue a simulation that has been interrupted before convergence, continue simulation with tighter convergence criteria, or create a project file for archive which can be rerun as a one-day transient simulation to quickly compute cyclic results.

Simulation Dates and Times

These dates and times are used to set the date and time labels put on steady state simulation results and to control the duration of transient simulations by allowing you to set the start and stop dates and times. Depending on the simulation method chosen different simulation "Dates and Times" fields will be enabled or disabled.

- **Steady & Cyclic Simulations:** For *steady simulations*, the date field is only used to label the simulation results, however, the time field is used to obtain schedule values from the day schedules based on the day type which the simulation is set to utilize. You set the *Day Type* via the steady state Weather and Wind Parameters property sheet (see Weather Properties under Working with Weather and Wind). ContamX will then utilize the *Day Type* and time to select schedule values for items that have schedules associated with them. The value of *Day Type* will be from 1 to 12 and will be used to select from the twelve day schedules associated with each week schedule in use during a simulation.

  For *cyclic simulations* with steady airflows, the date field is only used to label the simulation results, and the *day type* of the steady state data is used to determine schedule values. For cyclic simulation with transient airflows, the date field is used to determine schedule values and obtain data from weather and contaminant files if being used, otherwise steady state data is used. Cyclic simulations always begin at time 00:00:00 and end at time 24:00:00, so the time field is irrelevant.

- **Transient Simulation, Start and Stop:** These date and time fields are used to set the starting and stopping date and time of transient simulations. You should be sure to use only those dates available in both the weather and ambient contaminant files. The available dates in these files will be displayed in the Transient Weather Data and Transient Contaminant Data sections below. Each day of the WTH file contains a Day Type field which will be used to determine which of each week schedule's twelve day schedules should be used for each day of simulation.

  NOTE: If using steady state weather with a transient simulation, simulation will begin with the *Day Type* set in the steady state weather and cycle through the first seven day schedules if simulating more than one day.

  Use restart file: Each transient and cyclic simulation produces a restart file that contains the status of the simulation at midnight for each day of the simulation, and at the last time step of a simulation if it does not end at midnight. The restart file will have the same name as the PRJ file but have the RST extension. If a restart file with the same name as the project file is available, the dates for which restart data is available will be listed as "Restart data from ___ to ___" below the "Use restart file" check box. Enter the date in the edit box labeled “Restart data beginning.” If a restart file is used in a simulation, the existing restart file will be overwritten.

  This feature can be useful when running long multi-day simulations with the potential for generating impractically large detailed simulation results files. You can avoid the generation of large results files by running the multi-day simulation outputting only summary data files, reviewing the results to determine days of interest, e.g., highest concentration, then rerunning the simulation for the days of interest using the restart file and outputting detailed results.

  You can also use this feature to load *boundary layer and deposition/resuspension source/sinks* by first running a cyclic simulation for a single day, then using the restart file to run a transient simulation for the desired period of time.

Simulation Time Steps

These are the time increments that CONTAM uses when performing transient and cyclic simulations.

- **Calculation:** The *calculation time step* is that used by ContamX when performing transient airflow and contaminant calculations for the transient and cyclic simulations. Shorter time steps provide for greater accuracy of results but require more computation time. There must be an integral number of *calculation time steps* in an hour. The minimum allowable time step is 1 second and the maximum is 1 hour.
  Selecting a time step can also affect the ability to capture scheduled events. CONTAM provides the means to schedule events via day/week schedules and multiple time-based input files including WTH, CTM, WPC, CVF and DVF files. You can have ContamW help you choose this time step by selecting **Time Step Analysis...** from the **Simulation** menu. ContamW will review all schedules and input files to determine the maximum recommended time step required to capture all of the scheduled events. This will display a dialog box that will provide you with a list of

day schedules and input files along with the limiting time step for each and an overall limiting time step.

- **Output:** Simulation results are written to results files (see Working with Results) at the *output time step* which must be an even multiple of the *calculation time step*. Shorter time steps allow you to see the results in more detail, but also create larger results files. Utilizing an *output time step* that is longer than the *calculation time step* can still retain relatively high accuracy while reducing the size of the results files. Using these parameters you can refine your simulation period to provide detailed simulation results where needed.
- **Status:** CONTAM uses the *status time step* to determine how often the simulation progress is updated on the CONTAMX Simulation Control dialog box. The *status time step* must be a multiple of the *calculation time step*.

Simulation – Weather Properties

Steady State Weather Data: This is a summary of the steady state weather data settings. You can change this data by pressing the "Edit Weather Data…" button which will display the weather and wind related property sheet (see Defining Steady State Weather and Wind). You can have CONTAM use steady state weather data for both steady state and transient simulations. Note that a transient simulation that uses steady state weather or WPC file will still cycle through the first seven days of *schedules* beginning on the *day type* provided by the steady state weather data.

Transient Weather Data: Dynamic weather data can be in the form of a *CONTAM weather file* (WTH file) or a *wind pressure and contaminant file* (WPC file).

Select a WTH file by pressing the associated "Use WTH File…" button and finding the file you want to use during the simulation. Once you have selected a weather file, the date range for the particular file will be displayed in the Weather data from ___ to ___ fields. You can also check the "Use Steady State" button to reset the simulation to use the steady state data as opposed to a transient weather file. Note that if a WTH file is being used, then *schedule* values are determined from the *day type* data provided within the WTH file.

Transient Contaminant Data: Dynamic contaminant data can be in the form of a *CONTAM contaminant file* (CTM file).

Select an ambient contaminant file to be used by pressing the "Use CTM File…" button and finding the file you want to use during the simulation. Once you have selected a contaminant file, the date range for the particular file will be displayed in the Contaminant data from ___ to ___ fields. You can also check the "Use Steady State" button to reset the simulation to use the steady state data as opposed to a transient contaminant file.

Wind Pressure and Contaminant (WPC) Data: Transient weather and contaminant data can also be in the form of a WPC file. Click the "Use WPC File..." to browse for and select an existing WPC file (see Working with WPC Files).

A check box is provided to "**Only allow WPC and DVF files to trigger airflow calculations**." Normally, when using a WPC file, ContamX interpolates between entries of the WPC file if the timing of the entries in the WPC file does not match the simulation time step. This interpolation will typically lead to variations in wind pressures on the building envelope from one simulation

time step to the next. This in turn leads to the recalculation of the building pressures and airflows even if the interpolated WPC pressures are only slightly different from the previous time step. If working with very large project files, this can be very time consuming. Checking this box will allow for airflow calculations to be triggered only at those time steps for which there are corresponding entries in the WPC file that are different from those of the previously used entry of the WPC file. In a similar manner, an entry in the DVF file can be used to trigger an airflow calculation. Under this method of triggering airflow calculations, schedule and control values will be ignored except when an airflow calculation has been triggered. In essence, the airflow remains at steady state between WPC and DVF entries.

NOTES on Using WPC Files
Note that WPC files can be used during a simulation in conjunction with the transient weather and transient contaminant files as follows:

- If using wind pressures from a WPC file, you may not use a WTH file.
- If using contaminant concentrations from a WPC file, you may not use a CTM file.

If the simulation is set to use a WPC file then *schedule* values will be obtained by cycling through the first seven days of schedules beginning on the *day type* provided by the steady state weather data.

When performing a cyclic simulation and using a WPC file, the WPC file must contain only one day having the same date as indicated for *Steady & Cyclic Simulations*.

Simulation – Output Properties

When you run a simulation, results are saved to files as the simulation proceeds. Detailed results for airflows and/or zone contaminant concentrations are saved to the SIM file that is created when ever a simulation is run. The SIM file is a binary file which is not "human-readable" as opposed to human-readable text files. The data in the SIM file will be used by ContamW to display airflow results on the SketchPad, to display contaminant concentrations in the results display window, to create export and report files, and to generate charts of transient simulation results. Output is created at every *output time step*. Shorter *time steps* allow you to see the results in more detail, but lead to larger simulation result files. ContamX can create several result files in addition to the SIM file (See Working with Results), and you can control the data that will be saved during a simulation with the following parameters.

In most cases, ContamX can generate both **detailed** and/or **box-whisker** results. Detailed results are written to the corresponding file at each *output time step*. Box-whisker results include daily average, standard deviation, maximum and minimum values. Maximum and minimums are also accompanied by the time of day at which they occur for daily results, or the day on which they occur for the simulation summary results. Note that box-whisker data is based on calculations performed at the *calculation time step* but can require significantly less storage space. They can be of use when performing long simulations to find days and times of particular interest which can then be simulated in greater detail in conjunction with the *restart* (RST) file.

Airflow Simulation Results: These are the results available related to airflow calculations.

- **Airflow Rates:** Output airflows through and pressure differences across each flow link (airflow paths, duct segments, duct leakage paths, airflow terminals and simple air handler supplies and returns) to the SIM file. This data is necessary to display or

export results of airflow and pressures including those presented as SketchPad results and charts. You can also have these results written to a SQLite database file.

- **Building Air Change Rate:** Output the whole building air change rate to the ACH file. Select **Use Standard Density** to have ContamX use standard air density for the incoming outdoor air in the air change rate calculations, otherwise the outdoor air density will be based on the outdoor temperature.
- **Ages of Air:** Output age of air for each zone to the AGE file.
  WARNING: This calculation involves the inversion of the flow matrix and could be quite demanding on computational resources for projects involving a large number of zones.

Contaminant Simulation Results: These are the results available related to contaminant calculations.

- **Contaminant Concentrations:** Output contaminant concentrations of each contaminant node (zones and duct junctions) to the SIM file. This data is necessary to display or export results of contaminant concentrations including those presented as SketchPad results and charts. You can also have these results written to a SQLite database file. For 1D zones the average concentrations of all sub-cells will be saved to the SIM file. To obtain concentration time histories at specific locations in 1D zones, you can use zone sensors and report control nodes to output data to the *Controls LOG file* (see Results Files).
- **Occupant Exposure:** Output occupant exposure data to the EBW file. This data is necessary to display occupant exposure summary information when you highlight an exposure icon. You can still plot transient exposure data without this summary information.
- **Contaminant Exfiltration:** These files provide the amount of contaminant mass that is transported to the ambient. Two types of files can be created. The "Basic" file provides the total mass transported to ambient during each time step. The "Detailed" option creates a file for each contaminant and provides the mass of contaminant transported through each individual connection to the ambient. The basic file will simply have a CEX extension and the detailed files will have the contaminant name appended to the PRJ file name followed by the CEX extension.
- **Surface File:** The SRF file contains contaminant concentrations for source/sinks that include a reversible storage node. Unlike most source/sinks, these contribute a contaminant node to the system of contaminant equations. Currently there are two types: boundary layer diffusion and deposition/resuspension.

Short Time Step Results: These are the results when using the *Short Time Step* contaminant simulation method and the project contains one-dimensional convection diffusion zones (see Working with Zones). Four results files can be created, each having the same name as the PRJ file but with a different extension. Thes files are all binary files. NIST provides two utilities that you can use to "view" these files: ContamRV and 1DZRead. ContamRV is the results viewer that can provide a colorized view of the contaminant concentrations within the zones including the variations of concentrations within 1D convection/diffusion zones, and 1DZRead is a command-

line program that can convert the 1D results files to tab-delimitted files. File formats are provided in the Working with Results section.

- **RXR File:** This is a cross reference file that is created anytime at least one of the others is created.
- **RZF File:** This file contains the temperature, gage pressure and density of the zones.
- **RZM File:** This file contains the average mass fractions of each contaminants in each zone.
- **RZ1 File:** This file contains the mass fractions of each contaminant in each cell of each 1D zone.

Building Airflow Test Results: These options control which sections of data are written to the VAL file when you run the *Building Airflow Tests* simulation mode. See the Results Files section in the Working with Simulation Results section for detailed description of the VAL file.

- **Zones:** Write the "Zones" section that provides various air change rates for each zone to the VAL file.
- **Ducts:** Write the "Junctions" section that provides information related to the duct junctions and terminals to the VAL file.
- **Classified Flows:** Write the "Classified Zone Airflows" section to the VAL file. This section provides detailed mass flow rates to and from all zones categorized by various types of airflow sources, e.g., duct system, outdoor air, etc.

CONTAMX Display and Logging Options: These properties control the manner in which the simulation engine, ContamX, appears during execution and the amount of data written to the *CONTAMX.LOG* file during a simulation.

- **CONTAMX Display Mode:** Select whether you want to interact with ContamX via a Windows dialog box or Console window (DOS window). The dialog box method is the typical choice, but you might want to run it in a console window if you have trouble with ContamX executing properly particularly for large project files.
- **CONTAMX Log Mode:** Select the level of detail to output to the CONTAMX.LOG file. This is provided mainly for diagnostic purposes (for program developers) in the event that a problem is occurring during simulation (see Getting Help).
- **Echo .PRJ as read to .LOG:** Check this box to output the project file to the CONTAMX.LOG file. This is mainly provided for diagnostic purposes.

Simulation – Airflow Numerics Properties

These are the parameters that you can use to adjust the methods used to solve the airflow equations. Typically, you won't need to change these values from their defaults. However, certain types of problems may benefit from using different solution techniques that are available. For instance, large projects consisting of many zones and duct junctions could benefit from the use of the PCG linear equation solver.

Non-Linear Equation Solver: Calculation of airflows requires the solution of simultaneous non-linear equations. Two Newton-Raphson (N-R) algorithms are available: the **SUR** method

implementing under-relaxation the **Simple Trust Region** method. The latter method is more robust and is the default method. The first method is retained for compatibility with earlier versions of CONTAM. Convergence of the N-R solution of the non-linear simultaneous flow equations is controlled by four parameters:

- **Maximum Iterations:** Use this value to prevent endless calculations if the flow iterations are not converging. This should not be a problem with an appropriate relaxation coefficient.
- **Relaxation Coefficient:** This coefficient modifies the adjustment of the pure N-R method. Tests have found values near 0.75 to work well. Values near 1.0 may lead to non-convergent solutions. Values closer to 0.5 may be more reliable but take longer to solve. This coefficient is not used with the simple trust region method
- **Relative Convergence Factor:** The flow iterations are determined to have converged when the sum of all flows in and out of the zone are less than the *relative convergence factor* times the sum of the absolute values of those flows.
- **Absolute Convergence Factor:** A test is also provided in the event of very low flows to insure that the sum of the absolute values of the flows is less than the absolute convergence factor based on a very small zone air change rate entered here.

Linear Equation Solver: Select the method for solving the linear equations generated by the N-R method. ContamX uses either the **Skyline** algorithm (also called the profile method), which is a direct solution of the equations, or the Pre-conditioned Conjugate Gradient algorithm (**PCG**), which is the iterative biconjugate gradient algorithm. The execution time of the skyline method is determined by the sparsity and fill pattern of the solution matrix.

- **Maximum Iterations:** Use this value to prevent endless calculations if the flow iterations are not converging. This is only available for the PCG method.
- **Relative Convergence Factor:** The flow iterations are determined to have converged when the sum of all flows in and out of the zone are less than the *relative convergence factor* times the sum of the absolute values of those flows. This is only available for the PCG method.
- **Resequence Linear Equations:** The default is to resequence the linear equations in order to speed the solution by reducing the sparsity of the solution matrix. ContamX determines whether or not the resequencing actually improves the matrix. A failure in the resequencing algorithm is not a serious error because it will not change the numerical results. The resequencing uses algorithm 582, Collected Algorithms from ACM [ACM 1982].

  NOTE: If ContamX "crashes" during execution, it might work if you select not to resequence the equations or to select the *Console window* ContamX Display Option.
- **Linear Airflow Initialization:** An initial estimate of the pressures and flows is needed to start the non-linear equation solver. The linear initialization is usually faster than starting from zero.

Density Control Parameters: These parameters relate to the treatment of air density during simulations. Allowing for the variation of zone air density can provide transient analysis as

opposed to quasi-steady analysis that does not provide for non-flow processes to take place (see Basic Equations of the Theoretical Background section). Variation in zone density can be realized when using non-trace contaminants, e.g., water vapor, or when system flows are relatively large with respect to zone leakage. Typically, these parameters should be kept as their defaults, but they do allow for some more advanced treatment if desired.

- **Run steady state initialization to convergence:** A steady state simulation is always performed at the beginning of a transient simulation (unless using a restart file). If this box is checked, the flow calculation will be performed until the airflows, zone pressures and densities converge. If left unchecked, then the airflow calculation is performed without regard for changes in zone density due to pressurization/depressurization.

- **Vary density with zone pressure and Use advanced hydrostatic equation:** Zone density is computed using the gas law relationship $\rho = P / R \cdot T$, where by default, P is the barometric pressure. If this box is checked, then P is the actual zone pressure which is generally only likely to be a factor if the zone pressure varies significantly from barometric pressure.

  Typically the stack calculation is based on the incompressible hydrostatic equation of the form $P_s = -\rho \cdot g \cdot h$. The advanced equation accounts for the change in density with the local zone reference pressure as well, and has the form $P_s = P_{ref}(e^{-gh/RT}-1)$. These values are calculated for the inlet and outlets of each airflow path (and duct segment), and the difference is used to determine the contribution of the stack pressure to the overall pressure difference across the flow paths.

- **Vary density during time-step:** Allows the zone density to vary within the pressure calculation of each time step of a transient simulation until they have converged along with the pressures and airflow rates. This feature is not implemented when using the *short time step* method.

  Max Time Step Iterations: This is the maximum number of iterations for convergence of the density values.

- **Adjust temperatures in flow elements:** This 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.

- **Include dM/dt in transient simulation:** Set this option to account for transient flow processes that could account for accumulation or removal of significant amounts of air within a zone. This term is negligible for buildings with typical leakage and/or time steps greater than a few seconds.

Simulation – Contaminant Numerics Properties

These are the parameters that you can use to adjust the methods used to solve the contaminant equations. Typically, you won't need to change these values from their defaults. However, as discussed below, certain types of problems may benefit from using different solution techniques that are available.

Linear Contaminant Solver: Both the default solver and the CVODE solver require the use of a linear solution method. The need for a linear solver is described in the Contaminant Analysis

section. For each zone there is an equation for each contaminant species that you have selected to be either trace or non-trace contaminants. The mass fractions of the non-trace contaminants are solved within an iterative loop that also determines the airflows (because the non-trace contaminants affect the zone air densities and pressures). The mass fractions of the trace contaminants are solved after the flows have been computed (since they do not affect the zone air densities). These calculations require the solution of simultaneous, non-symmetric, linear algebraic equations.

Four linear solution methods are provided – two direct and two iterative. The iterative algorithms are successive over relaxation (SOR) and biconjugate gradient (BCG). The direct methods are a skyline (or 'profile') algorithm and a Gaussian LU-decomposition method. These four methods have been listed in order of increasing memory use. The iterative methods may or may not be faster than the more reliable direct methods. Before doing a long transient simulation of a large project, it may be useful to test the different methods to determine which gives optimum performance.

- **Skyline:** The skyline algorithm is the most reliable solution, and it is very fast for small problems but can be slow for large problems.
- **BCG:** Try the iterative biconjugate gradient algorithm for problems having a large number of zones for which the SOR algorithm might not converge.
- **SOR:** The successive over-relaxation algorithm requires much less memory and is faster than the Skyline method unless there are convergence problems.
- **LU:** LU decomposition using a full (non-sparse) matrix and is for verifying the solutions of the other methods rather than general usage.

  The following properties apply to the linear solver:

  - **Maximum Iterations:** Used to prevent endless calculations if the iterations are not converging.
  - **Convergence Factors:** The SOR iterations are assumed converged when the relative difference between the values on successive iterations are less than the relative convergence, e1. The BCG method is determined to have converged when $|A \cdot x - b| / |b| < e1$.
  - **Relaxation Coefficient:** An optimum over-relaxation coefficient, ω, speeds the convergence of the SOR algorithm. Determination of an optimum will require some experimentation with different values (usually near the low end of the range 1.0 to 2.0).
  - **The Trapezoidal Integration Factor:** Determines the relative portions of the explicit and implicit terms in the solution of the differential equations. This is the parameter γ presented in the Contaminant Analysis section of the Theoretical Background.
  - **Resequence Linear Equations:** The default is to do this in order to speed the solution by reducing the sparsity of the solution matrix.

CVODE Non-Linear Contaminant Solver: Select the method and associated parameters to use when implementing the CVODE solver.

- **Backward Differentiation Formulas**: The BDF method is recommended for stiff systems.
- **Adams-Moulton**: Suitable for non-stiff problems.

Maximum Time Step: CVODE limits its time steps according to its accuracy requirements, and in order to accommodate any scheduled events. This value limits the longest time step CVODE will take (the default value of 0 imposes no artificial restrictions). One reason to limit CVODE's time step is in order to allow the control network to respond to changes in contaminant concentrations. The control elements operate between steps of the contaminant transport solver, so if CVODE takes very long steps, it can interfere with simulating the building control system. Note that it is not necessary to set a maximum time step in order to ensure data get written to the simulation result files, because CVODE always interpolates its predictions to provide output at the requested times.

Relative Convergence Factor: Default value of 1×10^{-6}.

Absolute Convergence Factor: Default value of 1×10^{-13} kg/kg.

Convergence for Cyclic Simulations: Convergence is assumed when the relative difference in the peak concentrations (for all contaminants in all zones) is less than the relative convergence, e1. The absolute convergence, e2, is used to keep zones and contaminants with negligible mass fractions from dominating the solution.

1D Options: These options provide control over the 1D simulation capabilities that can be implemented when using the *Short Time Step Method* to perform contaminant simulation.

1D Zone Model: Select this method to implement the explicit one-dimensional convection/diffusion cell solution method for zones that have been defined as 1D zones. This will subdivide the zones into cells along user-specified 1D axes.

1D Duct Model: Select this method to implement the explicit one-dimensional convection/diffusion cell solution method for ducts. This will subdivide all duct segments into cells along their length based on the flow rate through the duct segment. Effective diffusion coefficient will be based upon the Reynolds number and the molecular diffusion coefficients of the contaminants.

If the product of the velocity u and the time step Δt in a given duct segment is less than the **Cell Size** parameter, then an Eularian method is used with a cell size equal to that of the *Cell Size* parameter, otherwise a Lagrangian method is implemented with an effective cell size of $u \cdot \Delta t$ leading to the solution of fewer simultaneous equations as the velocity increases.

Variable Junction Temperatures: When using the 1D Duct Model, you can have CONTAM implement variable junction temperatures to account for the convective heat transfer of the air within the duct system based on inlet terminal temperatures. When using this option, the junction temperature properties are ignored and terminal inlet temperatures are obtained from the zones in which they are located.

Simulation - CFD Numerics

These parameters are used to control the multizone/CFD coupling method. Theoretical background on the multizone/CFD coupling method is available in Wang [2007] and Wang and Chen [2007a]. Further background including validation and demonstration cases is provided in Wang and Chen [2007b], Wang [2008, 2010], and Wang, et al. [2010].

Number of Coupled Zones: This shows the number of CFD Zones in the project. This version of CONTAM only allows for one such zone.

Coupling Method:

- **CONTAM Only:** Perform only a multizone simulation - no CFD calculations will be performed.
- **CONTAM -> CFD:** Run multizone calculation then provide the results as boundary conditions for the CFD calculation. No information is passed back from the CFD calculation to the multizone calculation.
- **CONTAM -> CFD -> CONTAM:** The same as above, but the CFD results are passed back for use as boundary conditions for the multizone calculation. The data is exchanged in each *direction* only once.
- **CONTAM <-> CFD:** This is the fully dynamic coupling method. The multizone and CFD calculations provide boundary conditions iteratively to each other during each time step until either they reach convergence with each other at the multizone-CFD boundaries, or the *Maximum Coupling Iterations* is exceeded. The iteration process begins and ends with a multizone calculation.

Numerical Parameters:

Maximum Coupling Iterations: This is the maximum number of times, at each time step, that boundary conditions will be exchanged between the multizone and CFD calculations.

Output Frequency of Coupling Results: Controls how often the coupled simulation results are saved to the CMO file. This file is useful for monitoring the progress of the coupled simulation. These results are also output to the console during simulation.

Convergence Factor for Airflow Coupling: Convergence criteria for the dynamic coupling method.

Restart Coupling: Whenever a CFD airflow simulation is performed, a VAR file will be saved in the same location as the PRJ file. The VAR file contains the airflow results of the last time step for which a CFD calculation was performed. Check this option to run a contaminant simulation using the steady airflow results contained in the VAR file.

3.19 WORKING WITH RESULTS

CONTAM can provide you with several types of simulation results. The results available after a simulation depend on the simulation method and output parameter settings (See Output Properties in the Working with Simulations section). Once the results are available, you can view results using ContamW as described below or import files to a spreadsheet program for further analysis. Some files are created as text files while others are binary (non human-readable files). The text files can simply be imported into a spreadsheet program. The main results file is a binary file. ContamW provides the ability to generate tab-delimitted text files from this binary file which can then be imported into spreadsheets. In order to simplify access to the text-based results files, ContamW provides access to them via the File menu, and Windows allows you to create associations with the various file types. NIST also provides some post-processing tools to further manipulate the binary data and visualize results (see the Utility Software section of the NIST IAQanalysis website for details) .

3.19.1 Results Files

When ContamX performs a simulation, it creates a set of results files within the same directory that the project file is stored. These results files have the same file name as the project file with different extensions appended to indicate the type of results file. The following table lists the different file extensions and gives a brief description of the files. Details of each file are provided following the table.

Extension	Type	File Description
ACH	Text	Whole building air change rate - broken down into airflow path and ventilation system components and the combined value. This file can contain results for each time step and/or daily box-whisker results.
AGE	Text	Age of air - age of air results for each zone at each time step and/or daily box-whisker results.
BAL	Text	Balance - results of duct balancing calculation.
CBW	Text	Contaminant box whisker results - only created for daily box-whisker results.
CEX	Text	Contaminant exfiltration result.
CSM	Text	Contaminant source/sink summation - total mass of contaminants generated and/or accumulated by source/sinks and filters during a simulation.
EBW	Text	Occupant exposure results - average exposure of each occupant to each contaminant for each time step and/or daily box-whisker results.
LOG	Text	Controls log file - contains the output of all Report control nodes.
RST	Binary	Simulation restart file - only for use by ContamX for restarting simulations at the beginning of intermediate days of a simulation.
RXR	Binary	Short time step simulation results cross reference data
RZF	Binary	Short time step simulation results zone environmental properties
RZM	Binary	Short time step simulation results zone mass fractions
RZ1	Binary	Short time step simulation results 1D zone cell mass fractions
SIM	Binary	Detailed simulation results - airflow rates and pressure difference of flow links, reference pressures and temperatures of airflow nodes, and contaminant concentrations of contaminant nodes at each time step. This is the file from which SketchPad results and graphs are produced and displayed using ContamW. It is also the file from which Export and Report files can be generated.
SQLITE3	Binary	SQLite database file contains the same data as the SIM file.
SRF	Text	Surface result file - contains contaminant concentrations for source/sinks that include a reversible storage node.
VAL	Text	Model "validation" test results file - contains the results of either the Building Airflow Test or Building Pressurization Test airflow simulation methods.

<project name>.ACH - Whole Building Air Change Rate

The air change rate is calculated based on the summation of airflow rates from the ambient zone into the *conditioned space* of the building including ducts and simple air handlers. Results are presented based on total flows into the mechanical systems (ducts and simple AHS), airflow paths and total of all flows. The *conditioned space* includes all mechanical system volume and the volume of those zones which you select to *include in building volume* (See Zone Data in the Working with Zones section).

$$ACR = 3600 \frac{\sum \dot{m}}{\rho_{air} V_{bldg}}$$

where

ACR = whole building air change rate [h^{-1}]

$\dot{m}$ = mass flow rate into the conditioned space [kg/s]

ρ_{air} = density of incoming air [kg/m^3]

V_{bldg} = volume of conditioned space [m^3]

If there is a mass-imbalance for any airflow zone, ContamX will generate a warning message indicating the zone in question.

File Format (tab-delimited text):

The first line of the ACH file contains:

```
StartDate // first date of simulation (mm/dd → IX)
EndDate // last date of simulation (mm/dd → IX)
SimStep // simulation time step [s](IX)
achsave // save detailed results flag [0/1] (IX)
abwsave // save box-whisker results flag [0/1] (IX)
Vcond // volume of conditioned space [m^3] (R4)
```

If achsave is 1, then a header line is written for the detailed data:

```
"day  time  path  duct  total"
```

If abwsave is 1, then a header line is written for daily box-whisker data:

```
"day b-w avg dev min min max max"
```

Then for each day

If achsave is 1, then a line of data is written for each output time step for a day:

```
Date // date (mm/dd → IX)
Time // time (hh:mm:ss → I4)
ACRpath // air change rate via flow paths (R4)
ACRduct // air change rate via ducts (R4)
ACRtotal // ACRpath + ACRduct (R4)
```

If abwsave is 1, then three lines of daily box-whisker data are written - one for paths, ducts and total respectively. Each line contains:

```
date // date (mm/dd → IX)
type // "path" "duct" or "total"
avg // average value for day (R4)
dev // standard deviation (R4)
tmin // time at which min occurs (hh:mm:ss → I4)
min // minimum value for day (R4)
tmax // time at which max occurs (hh:mm:ss → I4)
max // maximum value for day (R4)
```

At the end:

If abwsave is 1, then three lines of summary box-whisker data for the entire simulation - one for paths, ducts and total respectively. Each line contains:

```
"final"
type // "path" "duct" or "total"
avg // average value for simulation (R4)
dev // standard deviation for sim (R4)
dmin // date on which min occurs (mm/dd → IX)
min // minimum value for simulation (R4)
dmax // date on which max occurs (mm/dd → IX)
max // maximum value for simulation (R4)
```

Example File:

```
1/1  1/1  3600  1  0  847.44
day  time  path  duct  total
1/1  01:00:00  2.911  0.010  2.921
1/1  02:00:00  2.932  0.098  3.030
```

<project name>.AGE - Age of Air

The mean age of air of a zone is an indication of the residence time of air within a zone. ContamX can calculate these values for all the zones in a project and provide them in detailed or box-whisker form. ContamX calculates these values using the method presented in AIVC Technical 34 [AIVC 1991].

File Format (tab-delimited text):

The first line of the AGE file contains:

```
StartDate // first date of simulation (mm/dd → IX)
EndDate // last date of simulation (mm/dd → IX)
SimStep // simulation time step [s](IX)
zaasave // save detailed zone age of air results flag [0/1] (IX)
zbwsave // save box-whisker results flag [0/1] (IX)
```

If zaasave is 1, then a header line is written for the detailed data:

```
"day time"  zone[] // …list of zone numbers
```

If zbwsave is 1, then a header line is written for daily box-whisker data:

```
"day  zone  avg  dev  min  min  max  max"
```

Then for each day

If zaasave is 1, then a line of data is written for each output time step for a day:

```
Date // date (mm/dd → IX)
Time // time (hh:mm:ss → I4)
age[] // age of air for each zone [1/hr]
```

If zbwsave is 1, then a line of daily box-whisker data is written for each zone. Each line contains:

```
date // date (mm/dd → IX)
zone // zone number (IX)
```

```
avg // average value for day (R4)
dev // standard deviation (R4)
tmin // time at which min occurs (hh:mm:ss → I4)
min // minimum value for day (R4)
tmax // time at which max occurs (hh:mm:ss → I4)
max // maximum value for day (R4)
```

At the end:

If zbwsave is 1, a line of summary box-whisker data is written for each zone for the entire simulation. Each line contains:

```
"final"
zone // zone number (IX)
avg // average value for simulation (R4)
dev // standard deviation for sim (R4)
dmin // date on which min occurs (mm/dd → IX)
min // minimum value for simulation (R4)
dmax // date on which max occurs (mm/dd → IX)
max // maximum value for simulation (R4)
```

Example File:

```
3 1/1 1/1 300 0 1
day zone avg dev min min max max
1/1 1 1.000 0.000 00:05:00 1.000 00:05:00 1.000
1/1 2 2.000 0.000 00:05:00 2.000 00:05:00 2.000
1/1 3 3.000 0.000 00:05:00 3.000 00:05:00 3.000
final 1 1.000 0.000 1/1 1.000 1/1 1.000
final 2 2.000 0.000 1/1 2.000 1/1 2.000
final 3 3.000 0.000 1/1 3.000 1/1 3.000
```

<project name>.BAL - Duct Balance Results File

This file contains the results of a duct balance operation if performed by ContamX. It contains the status of the balance operation along with detailed results and error tracking information if required.

File Format (tab-delimited text):

The first line of the BAL file contains the status of the balance procedure:

```
Code // balance completion code [0-5] (IX)
!Message[] // comment containing completion message (I1)
```

The next two lines are headers for fan and terminal data:

```
!fan duct # flow [kg/s] Prise [Pa] RPMratio
!trm jctn # rel-flow [-] Cb [-]
```

Then for each balancing terminal and associated fans:

If terminal:

```
"trm" // indicates terminal data
junc // junction number of terminal (IX)
rflow // relative flow Fbal/Fdes (R4)
```

```
 Cb // balance coefficient (R4)
```

If fan:

```
 "fan" // indicates fan data
 duct // duct segment number of fan(IX)
 flow  // total fan  flow (R4)
  prise // pressure rise across fan (R4)
```

If fan performance curve fan:

```
 frpm // RPM ratio required to achieve balance
 // flow rate with design fan (R4)
```

The last line is exactly:

```
end
```

Example File:

```
0 ! Flow balance iterations converged
!fan duct # flow [kg/s] Prise [Pa] RPMratio
!trm jctn # rel-flow [-] Cb [-]
fan 4 0.334472 68.0745
trm 1 1.000025 4.38419
trm 2 0.999974 4.2013
fan 8 0.334472 107.938 0.78357
trm 11 1.000007 7.76847
trm 12  0.999991 7.58544
end
```

<project name>.CBW - Contaminant Box Whisker

This file contains contaminant daily box-whisker results and summary box-whisker results for the entire simulation period. Results are presented for all zones within a project file as well as for each contaminant simulated.

File Format (tab-delimited text):

The first line of the CBW file is a header describing the amount of data in the file:

```
NumZones // number of zones (IX)
NumCont // number of contaminants (IX)
StartDate // first date of simulation (mm/dd → IX)
EndDate // last date of simulation (mm/dd → IX)
```

The second line is a header:

```
"day  zone  ctm  avg  dev  min  min  max  max"
```

Then daily box-whisker results for each day
for each zone
for each contaminant:

```
  date // date (mm/dd → IX)
  zone // zone number (IX)
  ctm // contaminant number (IX)
  avg // average value for day (R4)
  dev // standard deviation (R4)
```

```
min // minimum value for day (R4)
tmin // time at which min occurs (hh:mm:ss → I4)
max // maximum value for day (R4)
tmax // time at which max occurs (hh:mm:ss → I4)
```

After all daily values comes the summary for the entire simulation
for each zone
for each contaminant:

```
"final"
zone // zone number (IX)
ctm // contaminant number (IX)
avg // average value for simulation (R4)
dev // standard deviation for sim (R4)
min // minimum value for simulation (R4)
dmin // date on which min occurs (mm/dd → IX)
max // maximum value for simulation (R4)
dmax // date on which max occurs (mm/dd → IX)
```

Example File:

```
2  1  1/1  1/1
day  zone  ctm  avg  dev  min  min  max  max
1/1  0  1  0.000e+000 0.000e+000 00:05:00 0.000e+000 00:05:00 0.000e+000
1/1  1  1  1.036e-004 0.000e+000 00:05:00 1.036e-004 00:05:00 1.036e-004
1/1 2  1  6.839e-005  4.049e-005  00:05:00  0.000e+000  24:00:00  1.019e-004
final  0  1  0.000e+000  0.000e+000  1/1  0.000e+000  1/1  0.000e+000
final 1 1  1.036e-004  0.000e+000  1/1  1.036e-004  1/1  1.036e-004
final 2 1  6.839e-005  4.049e-005  1/1  0.000e+000  1/1  1.019e-004
```

<project name>.CEX - Contaminant Exfiltration

This file contains the mass of each contaminant that exits the building during each simulation time step.

File Format (tab-delimited text):

The first line provides the name of the project.

```
"Project:" PrjName[]
```

The next line provides a header.

```
"nctm units"
```

The next line provides the number of contaminant and the units of mass.

```
nc // number of contaminants (I4)
"kg" // mass units, kilogram
```

The next line is a header line.

```
"Date Time"
contaminant_name[nc] // name of each contaminant (I1)
```

For each time step a line is output containing:

```
Date // date (mm/dd → IX)
```

```
Time // time (hh:mm:ss → I4)
```

For each contaminant:

```
Mass // mass exiting to ambient during the time step (R4)
```

The final line provides the total mass of each contaminant that left the building during the simulation.

```
TotalMass[nc] // for entire simulation period (R4)
```

Example File:

```
Project: test-cex-a
nctm units
2 kg
Date Time CTM1 PART1
1/1 01:00:00 3.00001e-001 1.49983e-010
1/1 02:00:00 3.00001e-001 1.49983e-010
. . .
1/1 24:00:00 3.00001e-001 1.49983e-010
Sum 7.20003e+000 3.59958e-009
```

<project name>_<contaminant name>.CEX – Detailed Contaminant Exfiltration

These files provide the amount of a single contaminant that leaves the building through individual flow paths, duct terminals and duct leaks connected to ambient and for each simple air handling system exhaust. One file is generated for each contaminant simulated.

File Format (tab-delimited text):

The first line of the CEX file contains the name of the project.

```
PrjName[]
```

The next line is a header line.

```
"Date Time"
contaminant_name[nc] // name of each contaminant (I1)
```

The next line of the file provides general file information.

```
npath // number of connections to ambient
cont_name // name of contaminant in this file
"kg" // units of mass
```

The next line is the header line. The first column heading is date/time followed by a list of connections to ambient. Connections are provided with a type character followed by a number of the item according to the project file. Type characters are: 'p' = airflow path, 't' = duct terminal, 'l' = duct leak and 'a' = air handling system exhaust (e.g., p5 = airflow path 5 as found on the SketchPad).

```
"Date Time"
connection[nconn] // identifier of each connection to ambient
```

For each time step a line is output containing:

```
Date // date (mm/dd → IX)
Time // time (hh:mm:ss → I4)
```

For each connection:

```
Mass // mass exiting through given connection during time step (R4)
```

Example File:

```
Project: test-cex-a
npath ctm_name units
4 CTM1 kg
Date Time p1 p2 a1 t2
1/1 01:00:00 0.00000e+000 1.00000e-001 1.00000e-001 1.00000e-001
1/1 02:00:00 0.00000e+000 1.00000e-001 1.00000e-001 1.00000e-001
. . .
1/1 24:00:00 0.00000e+000 1.00000e-001 1.00000e-001 1.00000e-001
```

<project name>.CSM - Contaminant Source/Sink Summation

This file contains summary data related to contaminant simulation. This file is created automatically by ContamX whenever transient contaminant simulations are performed. Source/sink, filter and zone numbers are provided for referencing back to ContamW. The file consists of six sections:

Contaminant source/sink summary - This section provides the mass of contaminant generated, removed and stored for each source/sink during a simulation.

Filter challenge summary - This section provides the mass of contaminant to which each filter is exposed during a simulation (upstream challenge).

Filter loading summary - This section provides the mass of contaminant "loaded" onto each filter during a simulation (not including initial loading).

Filter breakthrough summary - This section provides indication of whether or not filter breakthrough applies. If breakthrough applies and occurs then the date and time when it occurs and the breakthrough efficiency are provided, otherwise the efficiency at the end of the simulation is provided. Breakthrough only applies to Simple Gaseous Filters for which a non-zero breakthrough efficiency is provided.

Interactions with ambient - This section provides the mass of each contaminant that enters and exits the building volume, i.e., those that are set to be included in the building volume.

Simulation time - This section provides information on the duration of the simulation.

File Format (tab-delimited text):

The first two lines of the CSM file display the PRJ file name and the date and time the simulation was run:

```
"Project:"  FilePath[]  // the full path of the PRJ file (I1)
"Time:"     SimTime[]   // PC time the simulation was started (I1)
-- Blank line --
This is followed by the number of contaminants simulated.
NumCont "contaminants" // number of contaminants (I4)
-- Blank line --
```

This is followed by the ***Contaminant source/sink summary section****.*
NOTE: the following section is only presented if source/sinks exist within the PRJ file.

The first line of the source/sink section is a header:

"Contaminant source/sink summary."

The next line contains the number of source/sinks:

```
NumSS "source/sinks" // number of source/sinks(IX)
```

The next line is a header:

```
s/s#  zn#  species  released  removed  stored  [kg]
```

Then a line of data for each source/sink:

```
  SSNum          // source/sink number (IX)
  Type[]         // source/sink type (I1)
  Name[]         // contaminant name (I1)
  ZoneNum        // zone number (IX)
  As             // surface area of deposition elements [m^2] (R4)
  mult           // multiplier (R4)
  Xmin           // location coordinates [m] (R4)
  Xmax
  Ymin
  Ymax
  Zmin
  Zmax
  MassRel        // mass released(R4)
  MassRem        // mass removed [kg] (R4)
  MassAcc        // mass accumulated including previous storage (R4)
-- Blank line ---
```

*This is followed by the **Filter challenge section**.*
NOTE: this section is only presented if filters exist within the PRJ file.

The first line of the section is a header:

```
"Filter challenge summary."
```

The next line contains the number of filters:

```
NumFilt "filters" // number of filters (IX)
```

The next line is a header:

```
  "path" Ctm[nc] "element" // Ctm[nc] is a list of contaminants
   // where nc is the number of contaminants.
```

Then two lines of data for each filter:
The first line is:

```
  FltNum     // number of flow component that contains the filter
       // and letter indicating type (p:path, d:duct, t:term)
  MassUp[nc] // challenge mass of each contaminant [kg] (R4)
  Name[]     // filter element name (I1)
```

The second line is:

```
  "--"    // indicates same filter as previous
  MassDown[nc] // unfiltered mass of each contaminant [kg] (R4)
  "downstream" // indicates downstream mass
-- Blank line --
```

*This is followed by the **Filter loading section**.*
NOTE: this section is only presented if filters exist within the PRJ file.

The first line of the section is a header:

```
"Filter loading summary."
```

The next line contains the number of filters:

```
NumFilt // number of filters (IX)
```

The next line is a header:

```
  "path" Ctm[nc] "element ns" // Ctm[nc] is a list of contaminants
       // nc is the number of contaminants.
```

Then a line of data for each filter:

```
  FltNum // number of flow component that contains the filter
   // and letter indicating type (p:path, d:duct, t:terminal)
  MassLoad[nc] // mass filtered during simulation [kg] (R4)
  Name[] // filter element name (I1)
  ns     // number of sub-elements, ns>0 for super-filter (I4)
```

If ns>0, then a line of data for each sub-filter of the super-filter:

```
  "--"    // indicates sub-filter
  MassLoad[nc] // mass filtered during simulation [kg] (R4)
  Name[]  // filter element name (I1)
  "0"      // zero sub-elements (I4)
-- Blank line --
```

*This is followed by the **Filter breakthrough section**.*
NOTE: this section is only presented if filters exist within the PRJ file.

The first line of the section is a header:

```
"Filter breakthrough summary."
```

The next line contains the number of filters:

```
NumFilt "filters"  // number of filters for which for which
     // breakthrough  is possible, currently on Simple
     // Gaseous types (IX)
```

The next two lines are headers:

```
"path element"
" contaminant date time efficiency"
```

Then a line of data for each filter:

```
  FltNum  // number of flow component that contains the filter
             // and letter indicating type (p:path, d:duct, t:terminal)
  Name[]  // filter element name (I1)
  nc      // number of contaminants (I4)
```

Then a line of data for each contaminant for which breakthrough is specified.

Each line contains:

```
  "--" // indicates continuation of filter
  Ctm[]  // contaminant name (I1)
```

If breakthrough occurred:

```
  Date[] // date on which breakthrough occurs (MMMDD → I4)
```

```
  Time  // time at which breakthrough occurs (hh:mm:ss → I4)
  bkeff  // breakthrough efficiency
```

If breakthrough does not occur:

```
  "--"  // indicates no breakthrough date
  "--"  // indicates no breakthrough time
  eff  // efficiency at end of simulation
-- Blank line --
```

*This is followed by the **Interactions with ambient section**.*
The next two lines of the section are headers. They are exactly:

```
"Interactions with ambient."
"contaminant flow from flow to [kg]"
```

Then a line of data for each contaminant:

```
  Ctm[]  // contaminant name (I1)
  MassFrom       // total mass entering from ambient zone [kg] (R4)
  MassTo         // total mass exiting to ambient zone [kg] (R4)
-- Blank line --
```

*The final section is the **Simulation time section**:*

It contains a header and three lines of data:

```
  "Simulation time:"
  tsec "seconds" // simulation time in seconds [s] (IX)
  thr  "hours,"  // simulation time in hours [h] (R4)
  tday "days"    // simulation time in days [days] (R4)
```

Example File:

```
Project: C:\CONTAM96\samples\.4\Filters\TestBK3.prj
Time: Mon Jul 18 10:17:22 2005

5 contaminants

Contaminant source/sink summary.
4 source/sinks
s/s# type species zn# As mult Xmin Xmax Ymin Ymax Zmin Zmax released removed
stored [kg]
1 ccf CO 3 0 1 0 0 0 0 0 0 7.20E-01 0.00E+00 0
2 brs Cl2 3 0 1 0 0 0 0 0 0 1.00E-04 0.00E+00 0
4 drs Cl2 4 0 1 0 0 0 0 0 0 0.00E+00 1.04E-01 0
3 bls Cl2 4 1 1 0 0 0 0 0 0 8.66E-03 8.67E-03 1.23E-05

Filter challenge summary.
3 filters
path CO  CO2 Cl2 part0.10 part1.00 element
6p 1.164e-001 1.829e+000 2.361e-001 1.204e-001 1.204e-001 Both
-- 6.215e-002 1.829e+000 1.079e-001 1.039e-001 7.389e-002 downstream
1d 1.164e-001 1.829e+000 2.361e-001 1.204e-001 1.204e-001 MERV10
-- 1.164e-001 1.829e+000 2.361e-001 1.039e-001 7.389e-002 downstream
2t 1.164e-001 1.829e+000 2.361e-001 1.039e-001 7.388e-002 GF0
```

```
-- 6.213e-002 1.829e+000 1.079e-001 1.039e-001 7.388e-002 downstream

Filter loading summary.
3 filters
path CO CO2 Cl2 part0.10 part1.00 element ns
6p 5.427e-002 0.000e+000 1.282e-001 1.649e-002 4.652e-002 Both 2
-- 0.000e+000 0.000e+000 0.000e+000 1.649e-002 4.652e-002 MERV10 0
-- 5.427e-002 0.000e+000 1.282e-001 0.000e+000 0.000e+000 GF0 0
1d 0.000e+000 0.000e+000 0.000e+000 1.648e-002 4.651e-002 MERV10 0
2t 5.427e-002 0.000e+000 1.282e-001 0.000e+000 0.000e+000 GF0 0

Filter breakthrough summary.
2 filters
path element nc
 contaminant date time efficiency
6p Both 2
-- CO Jan01 01:01:00 0.3
-- Cl2 Jan01 00:58:00 0.5
2t GF0 2
-- CO Jan01 01:01:00 0.3
-- Cl2 Jan01 00:58:00 0.5

Interactions with ambient.
contaminant mass from mass to [kg]
CO 0.000e+000 2.067e-001
CO2 0.000e+000 4.938e+000
Cl2 0.000e+000 3.011e-001
part0.10 0.000e+000 2.954e-001
part1.00 0.000e+000 2.414e-001

Simulation time:
7200 seconds
2 hours
0.0833333 days
```

<project name>.EBW - Occupant Exposure

Occupant exposure can only be calculated for transient contaminant simulations. The occupant exposure is calculated based on the contaminant concentrations within the zones (or zone cells when using 1D zones with the *short time step* method) occupied by each occupant. These zones (or cells) are determined based on the associated occupancy schedules. The exposure is determined by integrating the concentration vs. time using the mean-value of the contaminant concentrations to which the occupant is exposed during each simulation time step. The resultant integral is divided by the output time step then written to the file in units of kg_cont/kg_air.

File Format (tab-delimited text):

The first line of the EBW file is a header describing the data in the file:

```
NumExps // number of exposure icons (IX)
NumCont // number of contaminants (IX)
```

```
StartDate // first date of simulation (mm/dd → IX)
EndDate // last date of simulation (mm/dd → IX)
SimStep // simulation time step [s](IX)
expsave // save detailed results flag [0/1] (IX)
ebwsave // save box-whisker results flag [0/1] (IX)
```

If expsave is 1, then a header line is written for the detailed data:

```
"day  time  ctm"  exp[]  // exp[] is a list of exposure icon numbers
```

If ebwsave is 1, then a header line is written for daily box-whisker data:

```
"day  pexp  ctm  avg  dev  min  min  max  max"
```

Then for each day

If expsave is 1, then a line of data is written for each for each contaminant and output time step for a day:

```
Date // date (mm/dd → IX)
Time // time (hh:mm:ss → I4)
exp[] // exposure for each exp icon for timestep [kg/kg]
```

If ebwsave is 1, then a line of daily box-whisker data is written for each exposure icon:

```
  date // date (mm/dd → IX)
  exp // exposure icon number (IX)
  ctm // contaminant number (IX)
  avg // average value for day (R4)
  dev // standard deviation (R4)
  tmin // time at which min occurs (hh:mm:ss → I4)
  min // minimum value for day (R4)
  tmax // time at which max occurs (hh:mm:ss → I4)
  max // maximum value for day (R4)
```

At the end:

If ebwsave is 1, a line of summary box-whisker data is written for each exposure icon and for each contaminant for the entire simulation. Each line contains:

```
  "final"
  exp // exposure icon number (IX)
  ctm // contaminant number (IX)
  avg // average value for simulation (R4)
  dev // standard deviation for sim (R4)
  dmin // date on which min occurs (mm/dd → IX)
  min // minimum value for simulation (R4)
  dmax // date on which max occurs (mm/dd → IX)
  max // maximum value for simulation (R4)
```

Example File:

```
1  2  1/1  1/1  300  1  1
day  time  ctm  1
day  pexp  ctm  avg  dev  min  min  max  max
1/1  01:00:00  1  2.811e-007
```

```
1/1  01:00:00  2  9.970e-008
1/1  02:00:00  1  2.571e-007
1/1  02:00:00  2 6.109e-008
1/1  1  1  3.021e-007  0.000e+000  02:00:00  2.571e-007  00:05:00  3.592e-007
1/1 1  2  1.061e-007  1.945e-008  02:00:00  6.109e-008  00:05:00  1.562e-007
final  1  1  3.021e-007  0.000e+000  1/1  2.571e-007  1/1  3.592e-007
final  1  2  1.061e-007  1.945e-008  1/1  6.109e-008  1/1  1.562e-007
```

<project name>.LOG

If you perform a transient simulation and implement *report control elements*, ContamX will create a file in the same directory as your project file with the name of your project file and the .LOG extension appended (See Control Element Type: Report a Value). This file will contain a line of data at each o*utput time step* and a column of signal values for each *report control element*.

<project name>.RST - Simulation Restart File

Restart is an alternative to normal initialization of a simulation. It contains the state of all airflow and contaminant nodes, flow rates and pressure differences for all flow paths, contaminant storage terms for sinks, and signal values of various control node types. State data will be stored at 24:00:00 of all days simulated.

File Format (binary):

Header data:

```
 m[0] = 0L; /* TURBO C++ messes up the first bytes written; */
 m[1] = (I4)_nzone; /* therefore, send 4 unused bytes. */
 m[2] = (I4)_npath;
 m[3] = (I4)_nctm;
 m[4] = (I4)_njct;
 m[5] = (I4)_ndct;
 m[6] = (I4)_ncss;
 m[7] = (I4)_nctrl;
 m[8] = (I4)_rcdat.date_0;
 m[9] = (I4)_rcdat.date_1;
 m[10] = (I4)_rSizeData;
 m[11] = (not set)
m[1] thru m[7] allow ContamW to check for some changes in the project.
m[8] and m[9] are the date limits displayed to the user.
m[11] will allow reading all dates on the file -- could be used to create a
selection box of available dates.
```

RESTART DATA:

For all AF_NODEs:

```
R8 T - temperature
R8 P - pressure
R8 M - mass
R4 Mf[_nctm] - mass fractions
 For all AF_PATHs:
R8 Flow[0] - primary flow
```

```
R8 Flow[1] - secondary flow
R8 dP - pressure drop
```

For CSS_DSC:

```
CSE_EDS (I4)pss->local (stored as R4, converted to I4 in simulation)
CSE_BLS (R4)pss->local
```

For CT_NODEs:

```
SNSDAT: R4 oldsig
PICDAT: R4 oldsig, R4 olderr
HYSDAT: R4 oldsig
```

1D Simulation Results (RXR, RZF, RZM and RZ1)

The RXR file provides the number of contaminants, zones, and cells necessary to read the other three 1D simulation results files. 1D simulation results can be displayed and converted to tab-delimitted files using the utility programs provided by NIST: ContamRV and 1DZRead respectively.

These files require that the structures in ContamW and ContamX be compiled using no greater than 4-byte member alignment (under Visual C++). The file is unreadable if the default structure member alignment (8-bytes) is used.

The day and time values are provided so the program reading these files can insure it is reading from the proper position. The RXR file provides the number of contaminants, zones, and cells necessary to read the other three files.

RXR File Format (binary)

The first section of the file is:

```
24 // identifier (for ContamX 2.4) (I4)
_time_list // listing time steps [s](I4)
_date_0 // start of simulation - day of year (I4)
_time_0 // start of simulation - time of day [s] (I4)
_date_1 // end of simulation - day of year (I4)
_time_1 // end of simulation - time of day [s] (I4)
_urzf // if true, create RZF file (I4)
_urzm // if true, create RZM file (I4)
_urz1 // if true, create RZ1 file (I4)
_nctm // number of contaminants (I4)
_nzone // number of zones (excluding ambient) (I4)
_nZ1D // number of 1-D zones (I4)
_nC1D // total number of cells in 1-D zone (I4)
```

If _nZ1D > 0, then the following data is written for each 1-D zone:

```
nr // the ContamW zone number (I4)
nc // the number of cells in the zone (I4)
X1 // the x-coordinate of the axis origin [m] (R4)
Y1 // the y-coordinate of the axis origin [m] (R4)
Z1 // the z-coordinate of the axis origin [m] (R4)
X2 // the x-coordinate of the axis end [m] (R4)
Y2 // the y-coordinate of the axis end [m] (R4)
Z2 // the z-coordinate of the axis end [m] (R4)
```

RZF File Format (binary)

For each time step:

```
_dayofy // day of year [1 to 365] (I4)
_sim_time // time value [s] [0 to 86400] (I4)
for each zone:
T // Temperature [K] (R4)
P // Gage Pressure [Pa] (R4)
D // Density [kg/m3] (R4)
```

RZM File Format (binary)

For each time step:

```
_dayofy // day of year [1 to 365] (I4)
_sim_time // time value [s] [0 to 86400] (I4)
for each zone:
  CC[0] // mass fraction of species 0 [kg/kg] (R4)
  ...
  CC[n] // mass fraction of species n [kg/kg] (R4)
```

RZ1 File Format (binary)

For each time step:

```
_dayofy // day of year [1 to 365] (I4)
_sim_time // time value [s] [0 to 86400] (I4)
```

for each 1-D zone cell:

```
  CC[0] // mass fraction of species 0 [kg/kg] (R4)
  ...
  CC[n] // mass fraction of species n [kg/kg] (R4)
```

<project name>.SIM - Detailed Simulation Results

The SIM file is the main results file of ContamX containing detailed airflow and contaminant results. The data in the SIM file is used by ContamW to display airflow results on the SketchPad, to display contaminant concentrations in the results display window, to create export and report files, and to generate charts of transient simulation results. Output is created at every *output time step*. Shorter *output time steps* allow you to see the results in more detail, but lead to larger simulation results file.

The format of this file has been slightly modified from the previous version to accommodate a greater number of building components (zones, paths, ducts, etc.). This is reflected in the "nr" fields that have changed from I2 to I4. It is still a binary file (not "human-readable" as opposed to a text file) to provide for faster access and smaller file size than a text file.

Several post-processor programs are available from the NIST website (www.bfrl.nist.gov/IAQanalysis) that can be used to read the .SIM file and generate various forms of output from it. These programs include: SimRead2, SimReadW and ContamRV.

SimRead2 is a DOS-based command-line program that can be used to generate user-selected output from the SIM file. Output is in the form of tab-delimitted files. SimRead2 can be used in a

batch processing mode to minimize user-interaction when faced with the redundancy of manipulating multiple data sets.

SimReadW is a Windows program that can also generate user-selected output from the SIM file. SimReadW also provides the added capability of performing averaging of values over time and/or groups of building components, e.g., zones. It also enables the selection of either column-wise or row-wise orientation of the output.

The CONTAM Results Viewer, **ContamRV**, enables the visualization of contaminant concentrations on a color-coded image of the SketchPad. The program provides you with a color-coded display of zone contaminant concentrations, enables the animation of transient simulation results and displays zone concentrations in numerical format. A whole-building view is provided along with the ability to select individual levels of a project for detailed review.

File Format (binary):

The first 16 lines of the simulation results file contain data (32-bit integers) to help assure that the results apply to the project file currently in ContamW and to set the array sizes necessary to process the results.

```
24 // CONTAM version number  id (I4)
_nzone // number of airflow zones (excluding ambient) (I4)
_npath // number of airflow paths (I4)
_nctm // number of contaminants (I4)
_njct // number of junctions and terminals (I4)
_ndct // number of duct segments (I4)
_time_list // listing time steps [s](I4)
_date_0 // start of simulation - day of year (I4)
_time_0 // start of simulation - time of day (I4)
_date_1 // end of simulation - day of year (I4)
_time_1 // end of simulation - time of day (I4)
_pfsave // if true, write path flow results (I4)
_zfsave // if true, write zone flow results (I4)
_zcsave // if true, write zone contaminant results (I4)
_nafnd // number of airflow nodes (zones + junctions) (I4)
_nccnd // number of contaminant nodes (zones + junctions) (I4)
_nafpt // number of airflow paths (paths + ducts) (I4)
```

This is followed by _nafnd lines of airflow node cross-reference data:

```
  typ // source of node [zone, junction or terminal] (I4)
  nr // zone, junction or terminal number (I4)
```

The next _nafnd lines give the contaminant node cross-reference data:

```
  typ // source of node [zone or junction] (I4)
  nr // zone, junction or terminal number (I4)
```

The next _nafpt lines give the airflow path cross-reference data:

```
  typ // source of path [path, duct, or leak] (I4)
  nr // path, duct, or leak number (I4)
```

The simulation results for each day consist of:

The results for each time step consist of:

A line of time and ambient data:

```
dayofy // day of year [1 to 365] (I2)
daytyp // type of day [1 to 12] (I2)
sim_time // time value [s] [0 to 86400] (I4)
Tambt // ambient temperature [k] (R4)
P // barometric pressure [Pa] (R4)
Ws // wind speed [m/s] (R4)
Wd // wind angle [deg] (R4)
CC[0] // ambient mass fraction of species 0 [kg/kg] (R4)
...
CC[n] // ambient mass fraction of species n [kg/kg] (R4)
```

A line of data for each airflow path:

```
nr // path number; use as check (I4)
dP // pressure drop across path [Pa] (R4)
Flow0 // primary flow value [kg/s] (R4)
Flow1 // alternate flow value [kg/s] (R4)
```

A line of data for each airflow node (excluding ambient):

```
nr // node number; use as check (I4)
T // node temperature [K] (R4)
P // node reference pressure [Pa] (R4)
D // node air density [kg/m^3] (R4)
```

A line of data for each contaminant node (excluding ambient):

```
nr // node number; use as check (I4)
CC[0] // mass fraction of species 0 [kg/kg] (R4)
...
CC[n] // mass fraction of species n [kg/kg] (R4)
```

The time step data is followed by summary data for the day.

It begins with the following line of ambient data:

```
dayofy // day of year [1 to 365] (I2)
daytyp // type of day [1 to 12] (I2)
Tamax // maximum ambient temperature [k] (R4)
Tamin // minimum ambient temperature [k] (R4)
Pavg // average barometric pressure [Pa] (R4)
Wsmax // maximum wind speed [m/s] (R4)
Wsavg // average wind speed [m/s] (R4)
CC[0] // maximum ambient mass fraction of species 0 [kg/kg] (R4)
...
CC[n] // maximum ambient mass fraction of species n [kg/kg] (R4)
```

A line of data for each airflow path:

```
nr // path number; use as check (I4)
dPmax // maximum pressure drop across path [Pa] (R4)
Flowmax // maximum primary flow value [kg/s] (R4)
0.0 // place holder (R4)
```

A line of data for each airflow node (excluding ambient):

```
  nr // node number; use as check (I4)
  T // node temperature [K] (R4)
  P // node reference pressure [Pa] (R4)
  D // node air density [kg/m^3] (R4)
```

A line of data for each contaminant node (excluding ambient):

```
  nr // node number; use as check (I4)
  CCmax[0] // maximum mass fraction of species 0 [kg/kg] (R4)
  ...
  CCmax[n] // maximum mass fraction of species n [kg/kg]
```

Programmer's Note: this file requires that the structures in ContamW and ContamX be compiled using no greater than 2-byte member alignment (under Visual C++). The file is unreadable if the default structure member alignment is used.

<project file>.SQLITE3

The SQLITE3 file contains the same information as the SIM file but in SQLite3 database format consisting of the following tables: SimInfo, Time, AmbientConditions, NodeZoneFlowData, NodeZoneCcData, NodeDuctFlowData, NodeDuctCcData, LinkPathData, LinkDuctInfo, LinkDuctData. This file can be browsed and queried using a tool like *DB Browser for SQLite*.

<project file>.SRF

The SRF file contains contaminant concentrations for source/sinks that include a reversible storage node. Unlike most source/sinks, these contribute a contaminant node to the system of contaminant equations. Currently there are two types: boundary layer diffusion (BLS) and deposition/resuspension (DVR). The units for BLS nodes are kg of contaminant/kg of surface and those of DVR nodes are kg/m^2.

Surface Result File - File Format (tab-delimited text):

The first line of the file is a header providing the name of the PRJ file:

```
"Project:" file[] // PRJ file name (I1)
-- Blank line --
```

Then a header line:

```
"nr zone type species units trace/non-trace"
```

Then for each surface node :

```
  source    // number of source/sink (IX)
  zone      // number of zone in which source is located (IX)
  type[]    // type of source/sink ("bls", "dvr") (I1)
  species[] // name of species associated with source/sink (I1)
  units[]   // units of results (I1)
  trace[]   // type of source: "trace", "non-trace", "inactive" (I1)
-- Blank line --
```

Then a header line:

```
"date time " <type+nr> for each source/sink
```

Then a line of data for each listing time step:

```
  date   // date (mm/dd)
  time   // time (hh:mm:ss)
  conc[] // surface concentration for each surface node (R4)
```

<project file>.VAL

The VAL file contains the results of either the Building Airflow Tests or Building Pressurization Test airflow simulation methods. The file will be created or overwritten each time you perform one of these tests on a project file, so be sure to rename or copy any files you wish to keep.

If you perform a **Building Airflow Test** calculation, the data written to this file will be divided into five sections: Building info and ACH, Zones, Junctions, Classified Flows and Volumes. The first and last sections are always written, but the other three are controlled by the simulation output parameters for the Airflow Test simulation method.

Building Airflow Test - File Format (tab-delimited text):

The first line of the file is a header providing the name of the PRJ file:

```
"Airflow Summary for" file[] // PRJ file name (I1)
-- Blank line --
```

Weather conditions section:

```
"Ambient T:" Tambt "C"  // ambient temperature [C] (R4)
"Pressure:" Pbar "Pa"  // barometric pressure [Pa] (R4)
"Wind spd:" wspd "m/s" // wind speed [m/s] (R4)
"Wind dir:" wdir "deg" // wind direction [deg] (R4)
```

"Time:" time day[] // sim time (hh:mm:ss → I4) day of week (I1)

```
-- Blank line --
```

*Whole building air change rate - total flow from conditioned to unconditioned zones divided by the total volume of conditioned zones. Air change rate is calculated as ACH = F * 3600/(ZoneDensity * ZoneVol). ACH can be calculated from data provided in the Classified Flow section :*

```
"Bldg ACH:" ach   // building air change rate [1/hr] (R4)
-- Blank line --
```

Zones section:

This section is only written to the file if the simulation output parameter BldgFlowZ is equal to 1.

```
"Zones: nZones "flows [ACH]"// number of zones (I4) plus header
```

Data header:

```
zone C/U Supply Ret/Exh OA sys OA tot Circ tot P [Pa] T [C] Vol [m^3]
```

A line of data for each zone (excluding implicit zones of simple air handling systems, i.e., cu = S) - all flows provided as air changes per hour, ACH [1/hr]:

```
  nr  // zone number (I4)
  cu  // C:conditioned, U:unconditioned, A:ambt, S:AHS
  Qs  // Total supply airflow from system to zone.
      // Includes flow from: terminals, simple ahs, duct leaks
      // and forced flow paths
  Qrex // Total return + exhaust airflow from zone to system.
       // Includes flow to: terminals, simple ahs, duct leaks
       // and forced flow paths
  Qoasys // Total outdoor airflow from system to zone.
         // Includes flow from: terminals, simple ahs, duct leaks
         // and forced flow paths connected to Ambt
```

```
  Qoatot // Total outdoor air intake including all paths (non-forced flow)
           // connected to ambient: Qoasys + direct infiltration
  Qcirctot // "Circulation flow" sum of all flows out of zone
           // Includes flow to: terminals, simple ahs, duct leaks,
           // conditioned zones, unconditioned zones and Ambt
  P  // Zone pressure = absolute zone pressure minus the ambient
     // pressure at the reference elevation of the level on
     // which the zone is located [Pa]
  T  // Zone temperature [C]
  V  // Zone volume [m^3]
-- Blank line --
```

Junctions section:

This section is only written to the file if the simulation output parameter BldgFlowD is equal to 1.

```
 "Junctions:" nJct // number of junctions (I4)
```

Data header:

```
junction J/T  OA [%] Flow [kg/s] P [Pa] T [C] Vol [m^3]
```

A line of data for each junction:

```
  nr  // junction number (I4)
  jt  // J:junction, T:terminal
  poa // Percent outdoor air [%]
  w   // Airflow rate [kg/s]
  P   // Junction pressure [Pa]
  T   // Junction temperature [C]
  V   // Junction volume [m^3]
-- Blank line --
```

Flow classification section:

This section is only written to the file if the simulation output parameter BldgFlowC is equal to 1.

```
 "Classified Zone Airflows for " file[] ".prj" // PRJ file name (I1)
-- Blank line --
"Zones:" nZones "flows [kg/s]" // number of zones (I4) plus header
```

Data header:

```
"zone C/U Vol [m^3] Dens [kg/m^3] fmTerm fmLeak fmAHS fmFan fmCzone fmUzone
fmAmbt toTerm toLeak toAHS toFan toCzone toUzone toAmbt oaTerm oaLeak oaAHS
oaFan"
```

A line of data for each zone (excluding implicit zones of simple air handling systems, i.e., cu = S) - all flows provided in kg/s:

```
  nr  // zone number (I4)
  cu  // C:conditioned, U:unconditioned, A:ambt, S:AHS
  V   // Zone volume (volume for A is total cond volume
      // including zones, ducts and simple AHS) [m3]
  Dens    // Zone density [kg/m3]
  FfmTrm  // Airflow rate into zone from terminals
  FfmLeak // Airflow rate into zone from duct leaks
```

```
  FfmAHS  // Airflow rate into zone from simple AHS
  FfmFan  // Airflow rate into zone from forced flow elements
  FfmZnC  // Airflow rate into zone from conditioned zones
  FfmZnU  // Airflow rate into zone from unconditioned zones (except Ambt)
  FfmAmbt // Airflow rate into zone from ambient (except forced flows)
  FtoTrm  // Airflow rate out of zone to terminals
  FtoLeak // Airflow rate out of zone to duct leaks
  FtoAHS  // Airflow rate out of zone to simple AHS
  FtoFan  // Airflow rate out of zone to forced flow elements
  FtoZnC  // Airflow rate out of zone to conditioned zones
  FtoZnU  // Airflow rate out of zone to unconditioned zones (except Ambt)
  FtoAmbt // Airflow rate out of zone to Ambt
  FoaTrm  // Outdoor airflow rate into zone from terminals
  FoaLeak // Outdoor airflow rate into zone from duct leaks
  FoaAHS  // Outdoor airflow rate into zone from simple AHS
  FoaFan  // Outdoor airflow rate into zone from forced flow elements
-- Blank line --
```

Volume section:

```
"Volumes:"
"Conditioned zones"
VZCm "m^3"  // Total conditioned zone volume [m3]
VZCft "cuft" // Total conditioned zone volume [ft3]
"Ducts & AHS (conditioned)"
VDCm "m^3"  // Total system volume [m3]
VDCft "cuft" // Total system volume [ft3]
"Unconditioned zones"
VZUm "m^3"  // Total unconditioned zone volume [m3]
VZUft "cuft" // Total unconditioned zone volume [ft3]
```

Example File:

```
Airflow Summary for DEMO7_AirflowTest.prj

Ambient T: 20.00 C
Pressure:  101325.00 Pa
Wind spd:  0.00 m/s
Wind dir:  0.00 deg
Time:  00:00:00 Sunday

Bldg ACH: 0.450

Zones: 5   flows [ACH]
zone C/U Supply Ret/Exh OA sys OA tot Circ tot P [Pa] T [C] Vol [m^3]
1 C 0.78505 0 0.18038 0.18038 0.79 949.0 20.0 216.0
2 C 3.1401 0 0.7215 0.7215 3.14 949.0 20.0 54.0
3 C 1.5702 6.0457 0.36079 0.36079 7.85 949.0 20.0 108.0
4 C 9.4165 0 2.1636 2.1636 9.42 949.0 20.0 18.0
5 C 4.7083 0 1.0818 1.0818 4.71 949.0 20.0 36.0
```

```
Junctions: 12
junction J/T OA [%] Flow [kg/s] P [Pa] T [C] Vol [m^3]
1 T 23.0  0.056717 78.0 20.0 0.00
2 T 23.0  0.056715 63.3 20.0 0.01
3 T 23.0  0.056721 20.3 20.0 0.01
4 T 23.0  0.056692 0.0 20.0 0.01
5 T 100.0  0.065148 -0.0 20.0 0.00
6 J 23.0  0.28354 -0.2 20.0 0.11
7 J 23.0  0.28354 79.3 20.0 0.05
8 J 23.0  0.22682 64.5 20.0 0.06
9 J 23.0  0.17011 21.4 20.0 0.09
10 J 23.0  0.11338 1.1 20.0 0.07
11 T 0.0  0.21839 -880.2 20.0 0.04
12 T 23.0  0.056692 0.0 20.0 0.01

Classified Zone Airflows for DEMO7_AirflowTest.prj

Zones: 6   flows [kg/s]
zone C/U Vol [m^3] Dens [kg/m^3] fmTerm fmLeak fmAHS . . . oaLeak oaAHS oaFan
0 A 432.45 1.2041 0 0 0 . . . 0 0 0
1 C 216 1.2041 0.056717 0 0 . . . 0 0 0
2 C 54 1.2041 0.056715 0 0 . . . 0 0 0
3 C 108 1.2041 0.056721 0 0 . . . 0 0 0
4 C 18 1.2041 0.056692 0 0 . . . 0 0 0
5 C 36 1.2041 0.056692 0 0 . . .  0 0 0

Volumes:
Conditioned zones
432.00 m^3
15255.9 cuft
Ducts & AHS (conditioned)
0.45 m^3
15.8 cuft
Unconditioned zones
0.00 m^3
0.0 cuft
```

Building Pressurization Test - File Format (tab-delimited text):

The first line of the file is a header providing the name of the PRJ file:

```
"Building Pressurization Test for" file[] // PRJ file name (I1)
-- Blank line --
"Pressurization:"
Ptest "Pa"  // Test pressure (R4)
Ptest "in.H20"  // positive => pressurization
                // negative => depressurization
-- Blank line --
```

```
"Mass flow rate:"
F  "kg/s" // Flow rate required to obtain Test pressure (R4)
F  "kg/h"
F  "scfm"
-- Blank line --
"Volume flow rate:"
Q  "L/s"  // Flow rate required to obtain Test pressure (R4)
Q  "m^3/h"
Q  "cfm"
-- Blank line --
"Ambient conditions:"
"Pressure"
Pambt "Pa"  // Barometric pressure (R4)
Pambt "in.H2O"
"Temperature"
Tambt "C"  // Outdoor temperature (R4)
Tambt "F"
"Air Density"
Dambt "kg/m^3" // Outdoor air density (R4)
Dambt "lb/ft^3"
-- Blank line --
"Volumes:"
"Conditioned zones"
VZC "m^3" // Total volume of conditioned zones
VZC "cuft"
"Ducts & AHS (conditioned)"
VD  "m^3" // Total volume of ducts and AHS
VD  "cuft"
"Unconditioned zones"
VZU "m^3" // Total volume of conditioned zones
VZU "cuft"
-- Blank line --
"Airflow details:"
"Path Czone Uzone dP [Pa] [in.H2O] F [kg/s] [kg/h] [scfm] Q [L/s] [m^3/h]
[cfm]"
```

A line of data for each flow path contains airflow details. All flows and pressure differences are absolute values. If test is pressurization test, then all flows are from conditioned to unconditioned zones. If a depressurization test, then flows are from unconditioned to conditioned zones.

```
  nr  // path number
  Czone   // number of conditioned zone
  Uzone   // number of unconditioned zone
  dP_Pa   // pressure difference across path [Pa]
  dP_iH20 // [in.H20]
  F_kgs   // mass flow through path [kg/s]
  F_kgh   // [kg/h]
  F_scfm  // [scfm]
  Q_Ls    // volume flow through path [L/s]
```

```
  Q_m3h   // [m^3/h]
  Q_cfm   // [cfm]
```

Example File:

```
Building Pressurization Test for Test2_BldgPress_Auto.prj

Pressurization:
50.0 Pa
0.201 in.H2O

Mass flow rate:
0.4674 kg/s
1683 kg/h
822.6 scfm
Volume flow rate:
388.2 L/s
1398 m^3/h
822.6 cfm

Ambient conditions:
Pressure
101320 Pa
29.920 in.H2O
Temperature
20 C
68 F
Air Density
1.2040 kg/m^3
0.07517 lb/ft^3

Volumes:
Conditioned zones
445.44 m^3
15730.6 cuft
Ducts & AHS (conditioned)
17.11 m^3
604.3 cuft
Unconditioned zones
384.89 m^3
13592.3 cuft

Airflow Details:
Path Czone Uzone dP [Pa] [in.H2O] F [kg/s] [kg/h] [scfm] Q
[L/s] [m^3/h] [cfm]
3 4 1 49.95 0.2007 0.01077 38.78 18.95 0.008946 32.21 18.96
4 2 1 49.95 0.2007 0.009482 34.14 16.69 0.007875 28.35 16.69
5 3 1 49.95 0.2007 0.009482 34.14 16.69 0.007875 28.35 16.69
. . .
```

```
205 16 0 50.00 0.2009 0.009616 34.62 16.92 0.007987 28.75 16.92
206 16 0 50.00 0.2009 0.001923 6.924 3.384 0.001597 5.75 3.385
```

<project name>.XLOG – ContamX LOG File

The simulation engine will produce the text-based log file .XLOG within the same directory as the PRJ file. This file mostly contains simulation performance data that is typically not relevant to most users. However, if you encounter an error during simulation, this file can be useful to the program developers in tracing the error.

3.19.2 Viewing Results

This section describes how to activate the results display mode of ContamW and to display the results at different time-steps upon the SketchPad of transient simulations.

Results Display Mode

To view results using ContamW you activate the results display mode by checking either the **SketchPad Results** or the **Results Display Window** selection of the **View** menu. This will occur automatically after the successful completion of a simulation or after opening a project file that has simulation result files associated with it. Once the results display mode is activated, the status bar will display result-related information (See Status Bar) and you will be able to utilize all of the *Results Display Methods* outlined below.

Time Step

If you have performed a transient simulation, results will be available for each time step. However, the SketchPad results (as described in the following section) can only be displayed for a single time-step at a time. You can change the time-step for which results are currently displayed upon the SketchPad using the following menu and keyboard shortcut commands available under the **View** → **Results Navigation** menu.

Show Next Time Step
Menu Command: **Next Time Step**
Keyboard Shortcut: **End**

*Show Previous Time Step*Menu Command: **Previous Time Step**
Keyboard Shortcut: **Home**

Show Last Time Step (of the day)
Menu Command: **Last Time Step**
Keyboard Shortcut: **Ctrl+End**

Show First Time Step (of the day)
Menu Command: **First Time Step**
Keyboard Shortcut: **Ctrl+Home**

The increment for the following "Jump" commands can be set via the Results Display properties of the Project Configuration.

Jump Ahead (n time steps)
Menu Command: **Jump Ahead**
Keyboard Shortcut: **Shift+End**

Jump Back (n time steps)
Menu Command: **Jump Back**
Keyboard Shortcut: **Shift+Home**

Go to Date/Time…
Menu Command: **Go to Date/Time**
Keyboard Shortcut: **Ctrl+G**

3.19.3 Results Display Methods

This section describes the various methods that ContamW uses to present simulation results. These methods include SketchPad for airflow paths, a Results Display Window for more detailed information on the icon currently highlighted on the SketchPad, plotting transient results on graphs, shaft reports, and exporting results to various types of external files. You can also print copies of the graphs you view with ContamW or copy them to the Windows clipboard to be pasted into other applications (e.g. word processor applications for written reports).

3.19.4 SketchPad Results

ContamW will display color-coded bars indicating the relative airflow rates and pressure drops associated with each airflow path, duct terminal, or the supplies and returns of simple air-handling systems for the current time-step on the current level of the SketchPad. By default, **Airflow rates are displayed with green lines** and **pressure differences with red lines**. Display colors are user-selectable (See Configuring ContamW). These result-bars will only appear if you have selected the Airflows option on the Output page of the Simulation Parameters property sheet. If you have set flow or pressure limits for a given airflow path (see Airflow Path Properties), and they have been exceeded, the icon will be highlighted in red as a warning indicator.

You can also highlight individual icons on the SketchPad to display results related to the specific icon. Results can be viewed in the ContamW Status Bar and Floating Status Bar (See Status Bar in the Getting Started section of the documentation). The following is a brief explanation of the results displayed for each building component.

- Airflow path – pressure difference, airflow rate and direction of pressure drop and airflow
- Duct segment – pressure difference, airflow rate, velocity and direction of pressure drop and airflow where velocity is calculated from the area of the duct segment
- Simple air handling system – outdoor, recirculation and exhaust airflow rate
- Supply and return of simple air handling system – airflow rate
- Zone – zone temperature and reference pressure, i.e., the absolute pressure of the zone relative to the absolute pressure of the ambient at the elevation of the level on which the zone is located. Prior to CONTAM 2.4, this pressure was relative to the ambient pressure at the reference elevation of the building. So now these pressures more closely resemble a gage pressure.
- Duct junction – junction temperature and static pressure relative to the pressure at the height of the junction in the zone in which the junction is located

- Duct terminal – airflow rate and velocity based on the terminal free face area, temperature and static pressure relative to the pressure at the height of the terminal in the zone in which the terminal is located

3.19.5 Results Display Window

The ContamW Results Display window displays different types of information depending on the type of simulation that was performed (steady state or transient) the type of output that you selected (see Output Properties) and the currently (or most recently) highlighted icon on the SketchPad. The results display window will display information associated with zone icons and exposure icons. Once in the *results* mode, you can control the visibility of the Results Display window using the **Results Display Window** selection of the **View** menu. However, you can only display the Results Display window while the SketchPad results are displayed.

For steady-state simulations the steady-state results will be displayed. For transient simulations, results will be displayed for the current time step. Select the simulation time-step for which you want results to be displayed using the Simulation Time Step commands outlined in the previous section on Viewing Results.

Contaminant Concentration Results

If the currently highlighted icon is a zone icon, then contaminant results will be displayed in the Results Display Window. The concentration for each contaminant (species used in the simulation) will be displayed in the units of the default concentration of each contaminant (See Species Properties in the Working with Species and Contaminants section).

Zone Airflow Results

If the currently highlighted icon is a zone icon, then for each adjacent zone the sum of the airflows between the highlighted zone the adjacent zone will be displayed in the *Default Units of Flow* set for the project (See Default Units in the Configuring ContamW section).

Exposure Results

Summary exposure results are now available in the exposure box whisker (.EBW) file (See Results Files).

3.19.6 Plotting Results

ContamW provides you with some charting features for plotting transient simulation results. You can plot airflow, contaminant and exposure results using these features. Charting is only available for transient simulation results of the currently loaded CONTAM project. You can plot results for airflows and pressure differences, contaminant concentrations, and occupant exposure and potential dosage. You access these charting features from the **Simulation** menu when in the *results* mode.

NOTE: If you select a plot menu item when the ContamW caret is highlighting an icon, that item will appear in the list of items to plot when the associated Chart Control dialog box is displayed. This feature allows you to plot results for a particular item by highlighting it on the SketchPad as opposed to finding it in the list of items to plot on the Chart Control dialog box.

Plotting Airflow Results

Menu Command: **Simulation → Plot Airflow Results…**

Plot airflow and pressure difference results for airflow paths. You can plot two-way flow rates for those flow paths that implement two-way airflow elements. If you select not to show two-way flows, ContamW will plot the net flow rate for airflow paths that implement two-way airflow elements.

Plotting Contaminant Results

Menu Command: **Simulation → Plot Contaminant Results…**

Plot the time-history of contaminant concentrations of zones.

Plotting Exposure Results

Menu Command: **Simulation → Plot Exposure Results…**

Plot the time-history of occupant exposure to contaminants.

Plotting Particle Results

Menu Command: **Simulation → Plot Particle Results…**

Plot the time-history of the particle size distribution for a single zone.

Plotting Temperature Results

Menu Command: **Simulation → Plot Temperature Results…**

Plot the time-history of temperatures for multiple zones/junctions. This feature is provided for plotting temperature results as determined when performing coupled heat transfer calculations with either TRNSYS or EnergyPlus.

Chart Control: Transient Airflow Results

Use this dialog box to control how transient airflow and pressure results are plotted.

Paths to Plot/Paths Available: The plotted data for each airflow element will appear as a separate trace, the "Paths to Plot" lists the airflow elements that will be graphed. To add an item to this list select the appropriate airflow element type (paths, ducts, or junctions), scroll through the list of available elements or enter the element number directly if known, then press the "← Add" button. To remove an item from the Paths to Plot field select the zone to be deleted and then press the "Remove" button.

Date(s) to Plot: This field allows the you to select the dates of data you wish to plot on the graph. The available dates for the given simulation file are shown for your reference. You must set the dates to fall within this range of dates. Enter dates in the following format:

> MMMDD - the first 3 letters of the month and then the day of the month with or without spaces

Data to Plot: You can plot airflow rates, pressure differences, or both. If you plot both airflow and pressure difference, then ContamW will produce a graph with two vertical axis - one for airflow and one for pressure difference. You can select the units of these axis using the "Airflow Units" and "Pressure Units" drop-down combo boxes on this dialog box.

You can also show two-way flows for flow elements that utilize two-way flow models. If you leave "Show two-way flows" unchecked, then ContamW will plot the net flow rate for all flow paths. If you check "Show two-way flows," then a zero component will be plotted for all flow paths that do not implement two-way models.

Comment for chart title: Enter a comment you want to appear at the top of the chart to help distinguish various plots from one another.

Chart Control: Transient Contaminant Results

Use this dialog box to control how transient contaminant results are plotted.

Contaminant to Plot: This lists all the currently defined project contaminants. Select the contaminant for which you want results plotted.

Moisture Units: If the contaminant to plot is "H2O," then this section will be activated to provide the ability to plot moisture in units of *mass fraction*, *humidity ratio*, or *% relative humidity*. Mass fraction is the same units in which all other contaminants are reported, i.e., mass of contaminant per mass of air. Humidity ratio is in units of mass of water vapor per mass of dry air, i.e., mass of air less the mass of water vapor. *%RH* will utilize the equations presented in Chapter 6 of ASHRAE 2005 Fundamentals [ASHRAE 2005] to calculate the relative humidity based upon the mass fraction of H2O, temperature and absolute pressure in each of the *Zones to Plot*. If you select *Mass Fraction* or *Humidity Ratio* you can select from the set of units available in the Units list. When plotting humidity ratio, you should only plot in units of kg/kg, g/kg or lb/lb and not in volumetric units as the conversion will not be correct due to the assumed standard air density in CONTAM's conversion routines.

Zones to Plot/Zones Available: The plotted contaminant data for each zone will appear as a separate trace, the "Zones to Plot" lists the zones that will be graphed. To add an item to this list select the appropriate level and zone name from the drop down boxes in the Zones Available field and then press the "←Add" button. To remove an item from the Zones to Plot field first select the zone to be deleted and then press the "Remove" button.

Date(s) to Plot: This field allows the user to plot the contaminant results from a specified date range within the simulations results file. The available dates for the given simulation file are given, the user provided date range must fall between these dates. The dates are entered in the form: the first 3 letters of the month and then the day of the month with or without spaces.

Comment for chart title: Enter a comment you want to appear at the top of the chart to help distinguish various plots from one another.

Chart Control: Transient Exposure Results

Use this dialog box to control how transient exposure results are plotted.

Occupant to Plot: This is a list of the occupants for which you want to plot exposure results. Use the "Occupants Available" section to select occupants to add to this list and click the "←Add" button or highlight an occupant in this list and remove it using the "Remove" button.

Contaminant to Plot: This lists all the currently defined project contaminants. Select the contaminant for which you want results plotted.

Date(s) to Plot: This field allows the user to plot the contaminant results from a specified date range within the simulations results file. The available dates for the given simulation file are given, the user provided date range must fall between these dates. The dates are entered in the form: the first 3 letters of the month and then the day of the month with or without spaces.

Comment for chart title: Enter a comment you want to appear at the top of the chart to help distinguish various plots from one another.

3.19.7 Exporting Results

ContamW provides you with the ability to export results to external files. You can either generate *report files* or *export files*. *Report files* are text files that are formatted for ease of reading. *Export files* are *tab-delimited*, so you can easily import them into a spreadsheet. This feature provides you with greater charting flexibility as well as the ability to perform more sophisticated data analysis of your simulation results. You access this feature of ContamW by selecting **Export Results…** from the **Simulation** menu or use the **F5** shortcut key when the results-display mode is active. This will display the "Export Data" dialog box as described in the following section.

Export Data

This section provides detailed descriptions of the specific properties associated with generating result reports and exporting results to external files.

Transient Results:

Results available from/to: This shows the range of dates/times for which results are available in the simulation results file (.SIM) of the latest simulation.

Export transient results from/to: Select the range for the data to be exported. Enter the date as the first 3 letters of the month followed by the day of the month with or without spaces, e.g., Jan01, followed by the time in the following format HH:MM:SS.

Airflow Data: Options for reporting/exporting airflow data. Airflow rates will be reported in the *Default Units of Flow* as selected via the Project Configuration properties (See Configuring ContamW).

Report airflows of all paths for date/time: Create a text file formatted for ease of reading that lists the airflows and pressure differences for all paths, zone pressures and temperatures at the selected date and time.

Export airflows of all zones for date/time: Create a tab-delimitted file containing a table of the total airflows between each zone of the building. The leftmost column lists the level and name of the zones from which airflows and the topmost row lists the level and names of the zones into which air flows. Therefore, all airflows are listed as positive values.

Export transient flow for airflow path number: Create a tab-delimitted file listing all the airflow data for a specific airflow path. Select the airflow path number(s) for which you want to output the airflow and pressure drop data for the range of time you select in the "Export transient results from/to" dates and times indicated above.

Export average transient flow for each zone between export result times: Create a tab-delimitted file listing the average airflow rates between zones averaged over the range of time you select in the "Export transient results from/to" dates and times indicated above.

Contaminant Concentration Data: Options for reporting and exporting contaminant results. Concentrations will be reported in the units associated with the default concentration of each species/contaminant (See Species Properties).

Report transient concentrations of all zones for date/time: Create a text file formatted for ease of reading that lists the contaminant concentrations for all zones and contaminants at the selected date and time.

Report transient concentrations of all zones: Create a text file formatted for ease of reading that lists the contaminant concentrations for all zones at the selected date and time.

Export transient concentrations of all zones: Create a tab-delimitted file listing the transient contaminant concentrations for all zones and contaminants for the range of time you select in the "Export transient results from/to" dates and times indicated above.

Export transient concentrations for zone: Create a tab-delimitted file listing the transient contaminant concentrations for each contaminant in the selected zone for the range of time you select in the "Export transient results from/to" dates and times indicated above.

Shaft Report

CONTAM is well suited for the analysis of smoke control systems. Because of the importance of shafts for some smoke control systems [Klote and Milke 2002, Klote, et al. 2012], ContamW provides a special reporting feature for shafts.

Defining Shafts

You define shafts during the drawing phase of project development by placing zone icons directly below one another (in the same column and row) on adjacent levels of a building. If there is a zone icon in the same position on the level above or below the current level, those zones are part of the shaft when ContamW generates a shaft report. The shaft consists of zones that exist on contiguous levels having a zone icon in the same location on the SketchPad.

You create the shaft report by selecting **Generate Shaft Report** from the **Simulation** menu. This will display a dialog box prompting you to execute the following three steps to define the shaft zone icon and primary and secondary flow paths associated with the shaft across which you want ContamW to report the airflows and pressure drops. The zone icon would typically represent a stairwell, and the primary flow path would typically represent a stairwell door.

Steps to select a shaft for a shaft report:
1) Left-click on the zone icon that defines the shaft
2) Left-click on the primary airflow path icon
3) Left-click on the secondary airflow path icon
NOTE: Click the right mouse button to cancel the shaft report generation process.

Once you have completed the above steps, the shaft report will be displayed in a dialog box. The report will display pressure drop, airflow rates and direction of airflow on every level of the shaft where there are airflow path icons in the same location on the SketchPad as that of the primary and secondary airflow paths. You can use the Shaft Report dialog box to select the units in which the airflow rate and pressure drops are reported, and you can save a copy of the report to a text file.

3.20 WORKING WITH PROJECT ANNOTATIONS

Project *Annotations* and the *Project Description* provide you with simple project documentation features.

Annotations

You simply place an annotation icon onto the SketchPad (see Drawing Building Component Icons) and then use the icon definition procedure (see Defining Building Component Icons) to edit the text for the annotation. Once you have defined the text associated with the annotation icon, the text will appear in the status bar whenever you highlight the icon with the caret. You can edit, delete, and move the icon as you would any other building component icon.

Project Description

The project file description is provided to help you manage your project files. This description will be available when you are opening project files from within ContamW (see Working with Project Files).

4 SPECIAL APPLICATIONS OF CONTAM

This section includes instructions, or considerations, for using CONTAM to simulate several different building features (e.g. atria, stairwells and chimneys) and perform special building analysis such as fan pressurization. There are also available, several references that present the application of CONTAM in various studies [Emmerich and Persily 1996 and 1998; Emmerich and Nabinger 2000; Fang and Persily 1995; Musser and Yuill 1999; Musser 2000; Persily 1998; Persily and Martin 2000; Persily and Ivy 2001]. Also, check the bibliography provided on the NIST IAQ Analysis website http://www.bfrl.nist.gov/IAQanalysis/pubs.htm. Some documents are available directly from the website.

4.1 BUILDING PRESSURIZATION TEST

The building pressurization test (sometimes referred to as fan pressurization or blower door test) is a common way of determining the airtightness of a building envelope. Fan pressurization is relatively quick and inexpensive and it characterizes building envelope airtightness independent of weather conditions. In this procedure, a large fan or blower is mounted in a door or window and induces a large and roughly uniform pressure difference across the building shell [ASHRAE 2005, p 27.12; ASTM 1999]. A common airtightness rating is the airflow rate at 50 Pa divided by the building volume to give units of air changes per hour. CONTAM now provides an automated method of performing the *Building Pressurization Test* (See Working with Simulations), but there are also "manual" methods of performing this test presented below for informational purposes.

You may wish to do a simulated pressurization test to determine the airtightness rating of your idealized building. This is easily done by setting one of the zones on the zero elevation level to a constant pressure of 50 Pa, running the simulation, and then observing the total airflow out of that zone. (Don't forget flows to other building levels.) You could also define an airflow path using a constant volume airflow element through which you can vary the airflow until you achieve the desired level(s) of pressurization. Use standard m^3/h flow units and divide by the building volume to get the air change rate at 50 Pa. Alternatively, the flow rate at 4 Pa can be converted to an effective leakage area (ELA) using equations in Chapter 27 of the 2005 ASHRAE Fundamentals Handbook [ASHRAE 2005]. Use the CONTAM results display to check that the building has been properly pressurized by moving the cursor to each path on the envelope to get the pressure drop reported in the status bar. You can use the report of all flows to get a listing of all flow and pressure drops. The pressure difference (dP) across every path to the ambient zone should be close to -50 Pa. The value is negative, because the report considers flow and dP to the zone listed in the "zone" column from the zone in the "from" to be positive. All flows are to the "from" zone.

The reported pressure drop across various flow paths will not be exactly 50 Pa because of density effects. Minimize these differences by running all zones and ambient at the same (standard) temperature and no wind. Differences in pressure due to elevation make it critical that the constant pressure zone be on the level with zero elevation.

In a multizone building the effects of partitioning (internal walls and floors) usually are significant. There are two solutions depending on the goal of the simulation. If the goal is only to achieve a uniform pressure difference across the entire building shell, add a very large opening

across each partition. If the goal is to reproduce a field test, simulate the internal doors as being open to get a relatively uniform pressure inside the building.

Prior to performing these tests, you should make a copy of the project file, so you do not have to reverse the changes to restore the original project file.

4.2 Smoke Control Systems

For a stairwell pressurization system, you may want to record the pressurization flows by creating a simple AHS model with supply points in the shaft. Make these supply points the secondary path. You may use supply point icons aligned on every level with some or most of the flows set to zero, or you may use icons only on those levels with a pressurization flow. Alternatively, for shafts on an exterior wall of the building, you may wish to use the constant flow rate path elements through the exterior wall to provide the pressurization air. This type of flow path will also report the pressure difference between the shaft and ambient. You may want to align these constant flow paths above the open doorway on the first level. For details on viewing shaft reports see Working with Simulation Results in the Using ContamW section of the manual.

4.3 Shafts

Buildings contain several architectural features which offer very low resistance to airflow between building levels. These features include atria, elevator shafts, and stairwells. CONTAM provides two ways of modeling such a low resistance path: the phantom zone and the low-resistance (or large) opening. Select the most appropriate model based on how this path interacts with the rest of the model.

This interaction can be understood by using three flow resistances in series to model a shaft between two levels. There is a resistance to flow through the shaft and resistances representing the paths (doors) connecting the shaft to the two different levels. Using the powerlaw relationship, $Q = C(\Delta P)^{0.5}$, and assuming C = 3.10 for the shaft, 1.55 for an open door, and 0.01 for a closed door, the following equivalent flow coefficients, $Ce = (\Sigma 1/Ci^2)^{-0.5}$, are computed for four different assumptions:

1. both doors closed, ignore shaft resistance: Ce = 0.007070
2. both doors closed, include shaft resistance: Ce = 0.007066
3. both doors open, ignore shaft resistance: Ce = 1.096
4. both doors open, include shaft resistance: Ce = 0.984

Cases (1) and (2) show that when flow resistance is dominated by the closed doors, it is not necessary to include the shaft flow resistance. Cases (3) and (4) show that when the resistances at the (open) doors is of the same order as the shaft resistance, it is important to include the shaft resistance in the model. For a quick estimate of relative flow coefficients, remember that the flow coefficient is usually roughly proportional to the opening area.

4.3.1 Atria

If an atrium were modeled as individual zones on each level, the inter-level flow resistances would be very low. A cursory comparison of the inter-level opening areas to the sum of all openings from the atrium to other zones will usually indicate that an atrium should be modeled

using phantom zones. This is done by placing a standard zone icon at the atrium's lowest level. You use this zone icon to define the atrium temperature and total volume. You should ignore the floor area of the normal zone icon that is used to define the atrium zone. Place a phantom zone icon at all higher levels which contain parts of the atrium. A phantom zone icon indicates that the region within the walls on this level are actually part of the zone on the level below the icon. If it is necessary to have different temperature or contaminant concentrations at the different levels in an atrium, then those levels must be modeled as normal zones connected by flow paths with very large openings.

4.3.2 Stairwells

Data to describe the airflow characteristics of a stairwell in terms of an equivalent orifice is provided in [Achakji and Tamura 1988]. This representation is very well suited to being implemented by CONTAM, which allows you to create a powerlaw flow element from the physical characteristics of the stairwell. Stairwell characteristics include the cross-sectional (horizontal) area of the stairwell shaft, whether the fronts of the stair treads are open or closed, and the number of people on the stairs. This last item might be important in an evacuation scenario. Since the equivalent orifice area of the shaft is on the same order as it's cross-sectional area, and this area is likely to be smaller than the sum of the areas of the doors entering the stairwell, it is generally best to model a stairwell as individual zones on each level connected by openings which have been defined using the CONTAM stairwell airflow element that implements the Achakji/Tamura model.

4.3.3 Elevator Shafts

Elevator shafts (hoistways) will generally lie in some modeling regime that falls between stairwells and atria. Measured data for the flow resistance of an elevator shaft have not yet been identified. CONTAM provides a powerlaw flow element based on flow resistance from the Darcy/ Colebrook model of a conduit or duct. A significant difference between an elevator shaft and a stairwell is that the door between the shaft and the building is never fully open because, when it is open, air must flow through openings in the elevator car to pass between the building and the shaft. Table 6.1 of [Klote and Milke 2002] presents orifice areas for closed elevator doors which tend to be fairly small compared to the cross-sectional area of the shaft, especially for hoistways with multiple cars, so it is likely that the airflow in most elevator shafts can be modeled with sufficient accuracy by phantom zones. However, it may be necessary to split the shaft into several sections to achieve sufficient accuracy in modeling contaminant transport.

4.3.4 Chimneys

You can use a duct to create a simple chimney model. At its simplest this will involve only a single duct connecting the inside of the building to the outside with the Darcy-Colebrook duct element used to model the flow resistance of the chimney. The height of the duct terminal points must reflect the height of the chimney and a special high temperature zone must be created around the inside terminal point so that the proper stack effect will result. Be sure to allow an appropriate low resistance path from this special zone to the rest of the building.

5 WORKING WITH TRNSYS

The CONTAM simulation engine, ContamX, can be controlled via a set of TCP/IP socket communication messages. This enables CONTAM to be coupled with other simulation software. This socket communication capability, or bridge mode, has been utilized to couple CONTAM with TRNSYS (www.trnsys.com) to perform combined airflow and heat transfer simulations for multizone buildings. More specifically, TRNSYS implements a Type56 multizone building heat transfer module with which CONTAM can be "synchronized" to share inputs and outputs between a Type56 and a CONTAM representation of a multizone building.

Historically, TRNSYS has implemented Type96 and Type97, which were based on the airflow calculation capabilities of CONTAM and its predecessor program AIRNET [Walton 1989a]. These types only provided access to a subset of the airflow network calculations that CONTAM/AIRNET were capable of performing. Further, the types did not support the contaminant calculations that CONTAM was capable of performing. Therefore, a new Type98 has been developed that acts as a server to ContamX and provides communication and exchange of information/data between Type56 and ContamX. This implementation method provides the continued benefit of the full features of CONTAM as it evolves without requiring a redevelopment of a new type as previously required with Type96/97. In order to enable this coupling, new socket communications messages have been added. These are documented in Appendix B.

This section provides an overview of how to implement the latest version of Type98 [Dols 2014a, Dols 2014b] . This implementation is currently limited to the so-called *quasi-dynamic* (or ping-pong) method of coupling, whereby each analysis tool converges independently, as opposed to the *dynamic* coupling (or onion) method whereby convergence is attained between the coupled tools.

5.1 TRNSYS-CONTAM UTILITIES AND SETUP

Coupling between CONTAM and TRNSYS is further enabled using a set of NIST-developed tools that facilitate the process of developing the CONTAM and TRNSYS building models to be coupled and forming the connectivity between the inputs and outputs of the TRNSYS types. These tools are available on the IAQanalysis website (www.bfrl.nist.gov/IAQanalysis/index.htm) alongside of CONTAM. They are briefly described below along with instructions pertaining to where these files should be located on your Windows system in order to work with TRNSYS. In the file paths below, *Trnsys* refers to the directory into which TRNSYS was installed.

5.2 SOFTWARE COMPONENTS AND FILES

Type98<version>.dll – Type98 dynamic link library file developed to TRNSYS specifications. It is the go-between that handles communication of TRNSYS commands and data to/from ContamX. The “<version>” portion of the file name will depend on the versions of both TRNSYS and CONTAM for which the Type was compiled, e.g., “_t17_c32” indicates the DLL is compatible with TRNSYS version 17 and CONTAM version 3.2.

Copy to the *Trnsys\Userlib\ReleaseDLLs* directory.

ContamX3.exe – This is the calculation engine (solver) of the CONTAM program. It utilizes socket communication to transfer commands and data between Type98. It is installed in the CONTAM installation folder.

Copy to the *Trnsys\Exe* directory.

Type98.bmp – Type98 icon that can be associated with static-dynamic proformas created during CONTAM-TRNSYS coupling method outlined below.

Copy to the *Trnsys\Studio\Proformas\Utility\Calling External Programs\CONTAM* directory. This file is not critical, but usage is described in process below.

CONTAM3DExport.exe – This program enables the creation of a Trnsys3D file (.idf) by extruding CONTAM project files that have been drawn to scale using the new *pseudo-geometry* drawing features of ContamW.

An installer is provided by NIST, and the program can be accessed via the NIST folder of the Windows Start Menu.

Type56-98Coupler.exe – This program enables the creation of a static-dynamic proforma file (.tpf) based on CONTAM project files and TRNSYS Type56 building files as outlined in the coupling process below.

An installer is provided by NIST, and the program can be accessed via the NIST folder of the Windows Start Menu.

type98.log – This is a log file for Type98.

This file is created during a coupled simulation in the same folder as the associated PRJ file.

5.3 TRNSYS-CONTAM Coupling Process

The following outline and figure describe the process of coupling a CONTAM project via Type98 with TRNSYS Type56. It covers the basic method, but not all the details required for all possible coupling scenarios. You may need to implement scheduling, thermostatic control, etc. between the two types that are not automatically implemented by this process. However, this process does provide the connectivity between the zone airflows and temperatures. One should ensure that the CONTAM project to be used is fully defined prior to implementing the following coupling process, as it can be a bit cumbersome to modify significant features of the project once the coupling has been established. For example, adding/removing zones will cause major issues and require the coupling formation to be repeated. On the other hand, changing schedules, non-coupled controls, or adjusting simulation parameters of CONTAM should not require reformation of the coupling.

Automatic INPUTS to Type98

The coupling process below will automatically create TRNSYS INPUTS to Type98 for each zone temperature as well as ambient conditions: dry bulb temperature, barometric pressure, wind speed, and wind direction.

Manual INPUTS to Type98

To establish a manual INPUT, define a *Constant* control node in the CONTAM project file and give it a name. This can be used in many ways, e.g., to control the fraction of the design airflow rate of a supply point.

Automatic OUTPUTS from Type98

Each *connection* (which is made up of the sum of airflows) between two zones and between a zone and ambient is included as an TRNSYS OUTPUT from Type98.

Manual OUTPUTS from Type98

To establish a manual OUTPUT, define a *Signal Split* control node in the CONTAM project and give it a name.

Windows

To create a window for the purposes of thermal modeling via Type 56, define an airflow path within the CONTAM project that utilizes a two-way airflow element having a name that contains the word "wind". Place the airflow path icon on the wall where you would like the center of the window to be located. Define the elevation of the bottom of the window via the airflow path Reference Elevation and define the height and width of the window via the airflow element properties. If you do not want any airflow through the window during the coupled simulation, then set the path multiplier to zero. Upon export, the window will be created in the idf file according to these properties. You must ensure that the windows are fully contained within the wall and do not overlap each other.

The Coupling Process

1. Open/Create a project in **ContamW** sample project file name: [test.prj].
   The CONTAM model should be drawn to scale using a scaling factor. Using the scaling factor should result in 3D geometry for each zone that will produce a zone volume that closely matches the zone volumes.
2. Open the **CONTAM3DExport** program.
   a. Select the CONTAM project [test.prj]
   b. Click the *Save Trnsys3D IDF* button
      This will create a Trnsys3d compatible IDF file [test_T3D.idf]
   c. (Optional) Edit the IDF file with **Google SketchUp** using the **Trnsys3d** plugin. For example you may want to change the building to account for solar effects by changing the shape of a zone, e.g., the slope of the roof or add shading elements. You can also inspect the building model for problems with surfaces.
3. Open the **TRNSYS SimulationStudio** program
   a. Select the *File -> New* menu command
   b. Select *3D Building Project (multizone)* and click *Next >>*
   c. Select the idf file that was created in step 2 above [test_T3D.idf] and click *Open*. A prompt will appear to create a BUI file. This will also create a TRNSYS 17 building file [test_T3D.b17]. SimulationStudio will now display a Type56-based project in the Assembly Window.
   d. Save the TRNSYS project [test_T3D_imported.tpf]
   e. Exit **SimulationStudio**
4. Open the **Type56-98Coupler** program
   a. Select the CONTAM project file from step 1 above [test.prj].
   b. Select the B17 file produced in step 3 above [test_T3D.b17] and click *Open*.

c. Click the *Create Files* button to create an updated building file, a proforma and a variable exchange file. The updated building file [test_T3D_coupled.b17] has the following additions: airflow connections between zones, infiltrations added to zones with a connection to ambient, ventilation flows for duct terminals, AHS supply paths and duct leaks and inputs for these airflows. The proforma [test_T3D_coupled.tmf] is referred to as a dynamic-static proforma. It takes the place of the dynamic proforma for the previous Type 97. This is likely to be replaced in the future with a new dynamic proforma for Type 98. In the meantime, each project you create will implement its own dynamic-static proforma. The variable exchange file [test_T3D_coupled.vef] provides the list of variables to be used by Type98 to coordinate the data exchange during the simulation.

5. Establish the new proforma for use with TRNSYS

   a. Copy the proforma [test_T3D_coupled.tmf] to the following sub directory of the *Trnsys* directory: *Studio\Proformas\Utility\Calling External Programs\CONTAM*, or you can create your own directory for your CONTAM proformas under the *Proformas* directory.

   b. Additionally, you can copy the *Type98.bmp* file located in the *Trnsys\CONTAM* directory and give it the same name as that of the proforma but leave the *bmp* extension. This enables TRNSYS to display the new proforma in the Assembly Area with the Type98 icon.

6. Open the **TRNSYS SimulationStudio** program

   a. Open the *tpf* file that was created in step 3 above [test_T3D_imported.tpf]

   b. Change the Type56 unit to use the new building file from step 4 [test_T3D_coupled.b17].

      i. Double click the Type56 icon; select the *External Files* tab and *Browse* to select the B17 file.

      ii. The automatic OUTPUTS of Type98 should now be listed as INPUTS for Type56 under the *Inputs* tab. These are the inter-zone airflows or *connections* established for coupling. You should also see any manual outputs from Type98 that you created using ContamW in the list of available INPUTS.

7. Add the project-specific Type98 proforma to the TRNSYS project

   a. Select the newly created Type98 proforma [test_T3D_coupled.tmf] from the Direct Access Toolbar and drag it onto the Assembly Area.

   b. Edit the Type98 unit to set the PRJ and VEF file parameters to [test.prj] and [test_T3D_coupled.vef] respectively.

   c. The inputs and outputs listed in the VEF file [test_T3D_coupled.vef] should all be available via the Variables window of the unit (accessed by double clicking the icon).

8. Create a TRNSYS input deck file

a. The name of the deck file is determined by the *Global Infos* properties. You can change the name via the *Assembly -> Control cards…* menu command, scroll down to the *Deck file name* item and change the file name as desired. The default name is *default.dck*, but you can change it to anything you like, e.g., [test_T3D_imported.dck].

b. Select the *Calculate -> Create input file* menu command to create the deck file.

9. Update the TRNSYS deck file using the **Type56-98Coupler** program.

   a. Click the *Select the deck file* button and *Open* the deck file [test_T3D_imported.dck] that was created in step 8.

   b. If you closed the *Type56-98Coupler* program you also need to select the CONTAM project file again.

   c. Click the *Change Deck Name* button to set the name of the updated deck file [test_T3D_coupled.dck].

   d. Click the *Save New Deck File* button to create the new deck file. This deck file will have the connections between Type56 and Type98 established.

10. Import the new deck file back into the TRNSYS project.

   a. In **Simulation Studio** use the *File -> Import TRNSYS Input File…* menu command to open the newly created coupled deck file [test_T3D_coupled.dck].

   b. You should see the connections between Type56 and Type98 (test_T3D_coupled) and the INPUTS and OUTPUTS will be connected. If a weather file type reader, e.g., Type109 or Type15, is present, then that type will be coupled to Type98's ambient condition inputs. If there is more than one of these types then the first one found in the deck file will be used. Type15 is given preference over Type109, so Type15 will be coupled before any Type109. Be sure to select the proper file reader type for the weather file format being used, e.g., Type15-3 for EPW files.

   c. You can use the logging flag of the VEF file to provide various levels of details to the Type98.log file that will be created in the same directory as the TRNSYS project file. Set the value between 0 for no logging and 3 for detailed logging.

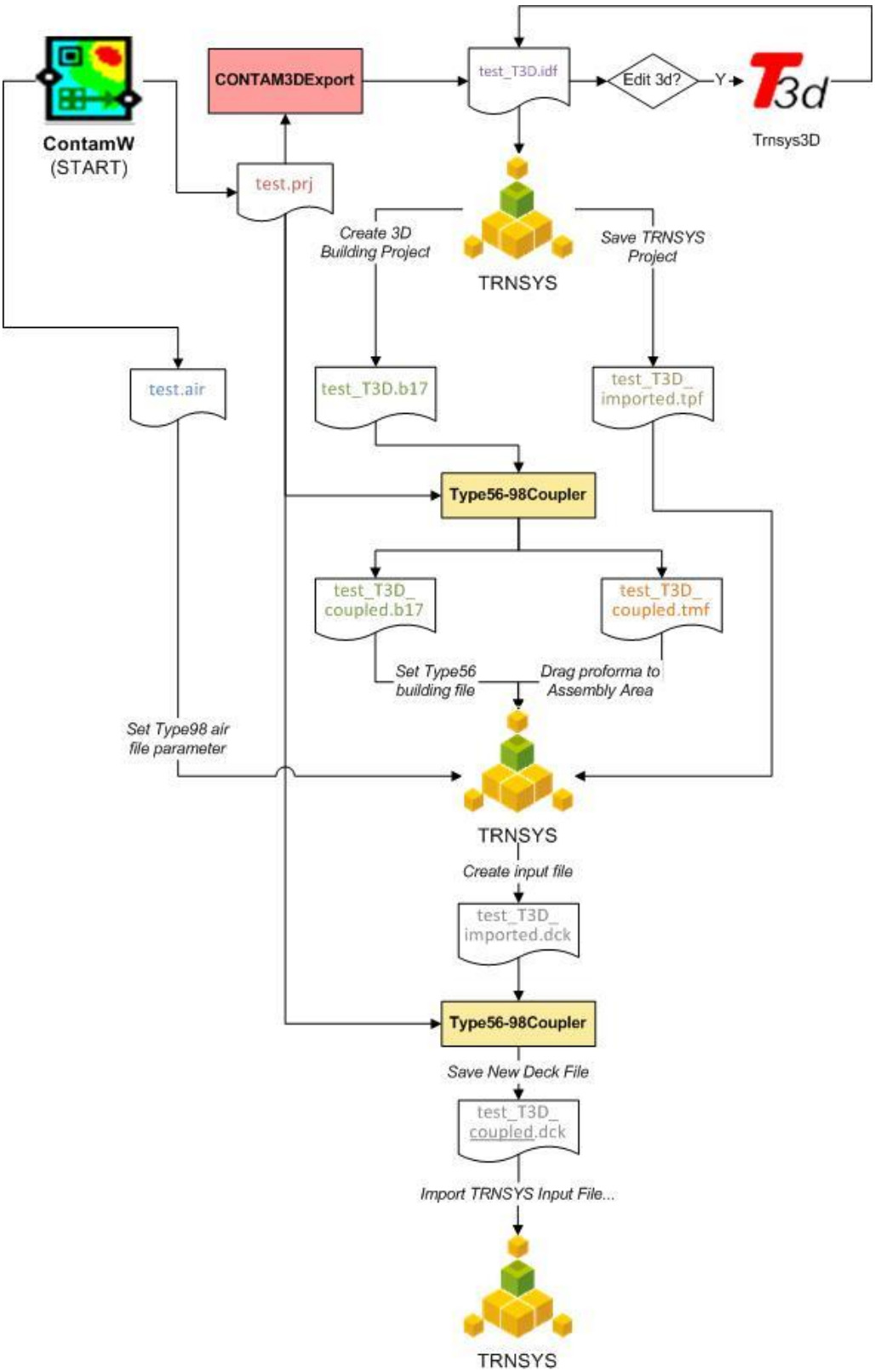
After following these steps you will have a TRNSYS project that has the temperature and flow connections made between Type56 and Type98 and possibly ambient condition connections between Type98 and Type15. You can then proceed with making any other desired connections, e.g., manual inputs/outputs established as described above.

NOTE: To run a simulation, the time step and the beginning and ending dates must be the same in both the CONTAM *project* file and the TRNSYS simulation control cards. ContamW can be used to view/plot temperature results.

TRNSYS-CONTAM Coupling Diagram

The following diagram illustrates the coupling process and the relationships between the files and software involved in the coupling process. This diagram is based on coupling the first version of Type98 that was developed for coupling CONTAM 3.1. The method has been slightly modified with the development of CONTAM 3.2 which utilizes the VEF file in place of the AIR

file. The VEF file can be generated using the Type56-98Coupler, and the content of this text file is also provided below. The above TRNSYS-CONTAM coupling process also includes the use of the VEF file.


ContamW
(START)
CONTAM3DExport
test_T3D.idf
Edit 3d?
Y
Trnsys3D
test.prj
Create 3D
Building Project
TRNSYS
Save TRNSYS
Project
test.air
test_T3D.b17
test_T3D_
imported.tpf
Type56-98Coupler
test_T3D_
coupled.b17
test_T3D_
coupled.tmf
Set Type56
building file
Drag proforma to
Assembly Area
Set Type98 air
file parameter
TRNSYS
Create input file
test_T3D_
imported.dck
Type56-98Coupler
Save New Deck File
test_T3D_
coupled.dck
Import TRNSYS Input File...
TRNSYS

TRNSYS Input Files

You can use ContamW to generate input data for the TRNSYS airflow module Type97 using the **Create TRNSYS (Type97) Input File** selection of the **Simulation** menu to generate ain AIR file. The data will be written to a file having the same name as the current project but the PRJ extension will be replaced with the AIR extension. Type98 has been developed to work directly with ContamX. Type 98 now uses a VEF (variable exchange file) instead of the AIR file. The VEF file is generated by the *Type56-98Coupler* utility as described in the TRNSYS CONTAM Coupling Process section.

Individual variables are briefly described in the following material. Units are indicated within brackets, [ ]. The type of variable is in parentheses.

`(I)` = integer value,

`(R)` = real/decimal value,

`(char)` = a single character

`(string)` = string of characters

Type98 VEF File

This *variable exchange file* is used by Type98 to coordinate data exchange information at the beginning of a simulation. This file will contain a list of variables that are to be exchanged during co-simulation. The number and types of variables will depend on the contents of the CONTAM project file.

File Format:

The first line is a file header indicates file type, version and logging flag:

```
VariableExchangeFile 1.0 trnsys 0 // set logging flag 0 - 3
```

The second line provides the number of variables being exchanged:

```
nv (I)
```

Each line will provide information related to a single variable to be exchanged during co-simulation:

```
io     // (char) "I" or "O" for input to/output from ContamX.
var_id // (string) - This identifier will be the name of the input or
       // output written to the dynamic/static proforma during the coupling
       // process.

var_type // (string) A three character abbreviation that indicates the type
         // of data that follows.
var_data // This information depends on the type as described below.
```

Type strings and related data:

- `cti` – *control input*. `CTRL_I_<name> <name>`
- `cto` – *control output*. `CTRL_O_<name> <name>`
- `zti` – *zone temperature input [C]*, `TAIR_<lev_nr>_<zone_nr> lev_nr zone_name`

- `wat` – *ambient temperature [C]*. `WTH_AmbientTemp`
- `wbp` – *barometric pressure [Pa]*. `WTH_BarometricPressure`
- `wws` – *wind speed [m/s]*. `WTH_WindSpeed`
- `wws` – *wind direction [deg]*. `WTH_WindDirection`
- `mix` – *interzone airflow rate [kg/h]*. `MIX_<lev_nr_from>_<zone_nr_from>_<lev_nr_to>_<zone_nr_to> lev_nr_from zone_name_from lev_nr_to zone_name_to`
- `inf` – *infiltration airflow rate [1/h]*. `INFIL_<lev_nr>_<zone_nr> lev_nr zone_name`
- `exf` – *exfiltration airflow rate [kg/h]*. `EXFIL_<lev_nr>_<zone_nr> <lev_nr>_<zone_name>`
- `sup` – *air handling system supply airflow rate [kg/h]* (output from ContamX). `AHS_supply_<path_nr> path_nr`
- `trm` – *duct terminal airflow rate [kg/h]*. `TERM_<lev_nr>_<zone_name> junction_nr`
- `dlk` – *duct leakage airflow rate [kg/h]*. `LEAK_<lev_nr>_<zone_name> junction_nr`

Type96/97 AIR File

The following CONTAM airflow elements are currently supported when creating TRNSYS input files. If any other elements exist within the current project, you will receive a warning message from ContamW that unsupported airflow elements exist within the project.

Powerlaw Models:

- $Q = C(\Delta P)^n$
- $F = C(\Delta P)^n$
- Leakage Area Data
- Connection (ASCOS) Data
- Orifice Area Data
- Crack Description
- Test Data (1-point)
- Test Data (2-points)
- Stairwell
- Shaft

Two-way flow model:

- Two-opening Model

In addition, the supply and return paths associated with the simple air handling system model are also included. The AHS implicit paths – recirculation, exhaust, and outside air – are not included. All the CONTAM building zones, AHS supply and return nodes, and the ambient zone are included. A complete list of the inputs to and the outputs from the TYPE96 subroutine are listed at the end of the .AIR file.

The inputs from TRNSYS to TYPE96 are:

- Temperatures for all zones including ambient [C]
- Absolute barometric pressure [Pa]
- Wind speed at the building reference elevation [m/s]
- Wind direction [degrees clockwise from north]
- All AHS supply and return flows [kg/s]
- Any other specified mass flow rates [kg/s]
- The relative areas of any variable openings

All flow and relative area inputs must be greater than zero.

The outputs from the TYPE96 subroutine to TRNSYS are the interzone air flows including flows to and from ambient. There are a maximum of two flows reported between any two zones − the sum of all flows from zone 1 to zone 2 and the sum of all flows from 2 to 1. There may be multiple paths between any two zones.

File Format:

`I` indicates an integer value and `R` indicates a four-byte real number

```
HEADER DATA:
Line 1: debug control
 I LIST - non-zero value initiates data dumps.
Line 2: counts
 I NWPP number of wind pressure profiles.
 I NZONE number of zones/simultaneous equations.
 I NPATH number of airflow paths.
 I NELMT number of airflow elements.
Line 3: run control
 I ITMAX - maximum number of iterations.
 R RELAX - under relaxation coefficient.
 R ACNVG1 - relative airflow convergence factor.
 R ACNVG2 - absolute airflow convergence factor.
 I TADJ - non-zero to do temperature adjustment.
Line 4: weather data
 R PBAR absolute barometric pressure [Pa].
 R WSPD wind speed [m/s].
 R WDIR wind direction [deg clockwise from north].
Line 5: end of section indicator
 I J - should equal -999

WIND PRESSURE PROFILES:
One line for each wind pressure profile consisting of
 I NR - profile number, starting at 1.
 I TYPE - profile type.
 I NPTS - number of data points.
 followed by one line with 5 real values for each data point.
```

```
Last line: end of section indicator
 I J - should equal -999

AIR FLOW ELEMENTS:
One line with 3 real values for each flow element.
Last line: end of section indicator
 I J - should equal -999

ZONE DATA:
One line for each zone consisting of
 I NR - zone number, starting at 1.
 I CP - 1 indicates constant pressure zone.
 R TZ - initial zone temperature [K].
 R PZ - initial zone pressure [Pa].
 R VOL - zone volume [m^3].
 R HT - zone elevation [m].
Last line: end of section indicator
 I J - should equal -999

PATH DATA:
One line for each airflow path consisting of
 I NR - path number, starting at 1.
 I NZ - number of zone air flows from (to be positive).
 I MZ - number of zone air flow to.
 R HT - path elevation [m].
 I TYPE - path type: 0=flow, 1 = PLR, 2=QCN, 3=FCN.
 I NE - number of the air flow element.
 R MULT - path multiplier.
 I XIN - index (>0) of XIN input value (type 0).
 I PWP - index (>0) of the wind pressure profile.
 R WIND - wind pressure modifier.
 R AZM - surface azimuth angle.
 R CTRL - flow or control value depending on type.
Last line: end of section indicator
 I J - should equal -999

INPUT & OUTPUT DATA:
First line:
 I NIN number of input variables.
 I NOUT number of output variables.
One line for each output variable consisting of
 I NR - output number (index to XOUT).
 I NZ - number of zone the air flows from.
 I MZ - number of zone the air flows to.
Last line: end of section indicator
 I J - should equal -999
```

6 WORKING WITH ENERGYPLUS

Beginning with CONTAM 3.2, ContamX can be coupled with EnergyPlus to perform combined airflow and heat transfer simulations. This coupling is performed using CONTAM's previously developed inter-process communication application programming interface (API). This API consists of a set of socket communication messages that enable the control of execution and exchange of data between ContamX and an external process. This same socket communication capability, or bridge mode, has been previously utilized to couple CONTAM with TRNSYS (www.trnsys.com) as documented in the Working with TRNSYS section. The socket communication API is documented in Appendix B. Coupling between ContamX and EnergyPlus is accomplished via co-simulation based on the Functional Mock-up Interface (FMI) for Co-simulation specification version 1.0 [MODELISAR 2010]. EnergyPlus provides a master algorithm that controls the simulation [DOE 2013, Nouidui et al. 2014, Wetter 2011], and a newly developed module (ContamFMU.dll) is incorporated into a Functional Mock-up Unit (FMU) for use during execution.

EnergyPlus sets up the dynamic data exchange between itself and the slave FMU based on the co-simulation description file. Currently, the following data can be exchanged between EnergyPlus and ContamX:

- From EnergyPlus to ContamX
  - **Zone Temperatures**
  - **Ventilation system airflow rates** for zone supply and return airflow rates
  - **Outdoor airflow fractions** of outdoor airflow controllers
  - **Exhaust fan airflow rates**
  - **Energy recovery fan airflow rates**
  - **Outdoor environment data** including temperature, barometric pressure, wind speed and wind direction obtained from EnergyPlus weather (.epw) files
  - **Output variables** which may be EnergyPlus Output:Variable or EnergyManagementSystem:OutputVariable objects
- From ContamX to EnergyPlus
  - **Zone infiltration airflow rates**
  - **Inter-zone airflow rates** via airflow paths, i.e., not via ducts
  - **Controls values** which may be any value exposed via a named “split” type control node in a CONTAM controls network, e.g., a signal calling for ventilation airflow due to an elevated contaminant level

The CONTAM3DExporter tool was developed by NIST to assist in the coupling between CONTAM and EnergyPlus. This tool reads a CONTAM PRJ file and creates an IDF file that contains objects associated with the geometry, infiltration and inter-zone mixing flows, an air loop that includes an outdoor air mixer and HVAC airflow terminal, and FMI-related entities required to facilitate dynamic data exchange. Detailed mappings between CONTAM and EnergyPlus entities are provided in the table below. Items in the table with fixed-width fonts,

e.g., `ZoneMixing`, indicate entities that are objects within CONTAM or EnergyPlus. Italicized items indicate EnergyPlus variable names made accessible via the EnergyPlus external interface, e.g., *Zone Mean Air Temperature*. The Notes column provides details of the relationship between the CONTAM and EnergyPlus entities.

CONTAM	EnergyPlus	Notes
Building Geometry		
`Zone`	`Zone`	CONTAM airflow zone maps to EnergyPlus thermal zone
Wall	BuildingSurface:Detailed	CONTAM zones are bounded by implicitly defined walls
Infiltration and Inter-zone Airflows (paths are grouped into connections between zones including ambient)		
`Airflow Path` (external)	ZoneInfiltration:DesignFlowRate	EnergyPlus constant term coefficient is set to 1 and all others to 0
`Airflow Path` (inter-zone)	ZoneMixing	Two ZoneMixing objects are created for each inter-zone connection
Ventilation System		
`Simple Air Handling System` (AHS)	AirLoopHVAC	Zone Supplies and Returns must exist in pairs to match the EnergyPlus air loop requirements
	Fan:ConstantVolume	
	AirLoopHVAC:ZoneSplitter	
	AirLoopHVAC:ZoneMixer	
	AirLoopHVAC:OutdoorAirSystem	
	`Branch` and `Nodes`	Outdoor air, Supply-side, Demand-side, etc.
AHS Supply	AirTerminal:SingleDuct:Uncontrolled	
	AirLoopHVAC:SupplyPath	
	Node	Zone supply air
AHS Return	AirLoopHVAC:ReturnPath	
	Node	Zone return air
FMI-related External Interface		
ExternalInterface:FunctionalMockupUnitImport:From:Variable (Values passed FROM EnergyPlus TO ContamX)		
	Output: Variable Index Key Name	
Environment	*Drybulb temperature, Barametric Pressure, Wind Speed and Direction*	
Zone	*Zone Mean Air Temperature*	
Supply Air Node	*System Node Mass Flow Rate*	
Return Air Node	*System Node Mass Flow Rate*	
AHS	*Air System Outdoor Air Flow Fraction*	
`Control` (Constant type)	(user-defined)	User must define connectivity after export
ExternalInterface:FunctionalMockupUnitImport:To:Actuator (Values passed TO EnergyPlus FROM ContamX)		
	Actuated Component Type	
`Airflow Path` (external)	*Zone Infiltration*	Multiple external airflow paths for a zone are grouped together into a single connection
`Airflow Path` (inter-zone)	*ZoneMixing*	Multiple interzone airflow paths for a pair of zones are grouped together into a single connection
`Control` (Signal Split type)	(user-defined)	User must define connectivity after export

6.1 ENERGYPLUS-CONTAM COUPLING PROCESS

The following outline describes the process of coupling a CONTAM project with EnergyPlus. It covers the basic method, but not all the details required for all possible coupling scenarios. This process utilizes the *CONTAM3DExport* tool (referred to as the exporter) to generate an EnergyPlus IDF file from a CONTAM PRJ file.

It is not absolutely necessary to follow this process in order to develop a coupled project. One could develop a CONTAM project with corresponding thermal zones to an existing EnergyPlus IDF, generate an IDF file with the CONTAM3DExport tool, and merge parts of the generated IDF file into the EnergyPlus IDF file. However, coupling without this procedure will require care in providing subsurfaces to contain individual flow paths of the associated CONTAM project. Also, air loops and nodes of the generated and EnergyPlus IDF files will need to be coordinated.

Create the CONTAM Project using ContamW

Use the following guidelines when creating your CONTAM project to be coupled with EnergyPlus. The table in the Working with EnergyPlus section shows the objects created in the EnergyPlus model and the corresponding CONTAM entities.

Geometry

Utilize the *pseudo-geometry* mode of ContamW in order for CONTAM3DExport to generate the necessary geometry of the EnergyPlus model. For each zone in the CONTAM model a thermal zone will be created in the EnergyPlus model.

Airflow Paths

Airflow paths will be grouped to form inter-zone mixing and infiltration airflow rates in the EnergyPlus model.

Windows

To create a window for the purposes of thermal modeling via EnergyPlus, define an airflow path within the CONTAM project that utilizes a two-way airflow element having a name that contains the character string "**wind**" (not case-sensitive). Place the airflow path icon on the wall where you would like the center of the window to be located. Define the elevation of the bottom of the window via the airflow path Reference Elevation, and define the height and width of the window via the airflow element properties. If you do not want any airflow through the window during the coupled simulation, then set the path multiplier to zero. Upon export, the window will be created in the IDF file according to these properties. You must ensure that the windows are fully contained within the wall and do not overlap each other.

Simple Air Handling Systems

An air loop will be created in the EnergyPlus model for each simple air-handling system (AHS) in the CONTAM model except as noted below for the exhaust-only and heat recovery ventilator (HRV) systems. In order to adhere to the EnergyPlus air loop paradigm, each thermal zone that is served by the air loop should have corresponding supply and return air terminals served by the associated "normal" simple AHS in the CONTAM building model. A zone can only be served by one air loop, so no more than one normal AHS should serve a zone in CONTAM. However, a zone can be served by one of each type of AHS: normal, exhaust-only and HRV; as described below.

Simple AHSs in CONTAM can be made to correspond to exhaust-only and energy recovery systems in EnergyPlus. The name of the simple AHS is used to trigger CONTAM3DExport to

create these systems along with system-specific supply and return terminal requirements. Exhaust-only system names must contain "**exh**" (not case-sensitive), and should not have any supply air terminals associated with them. Energy recovery system names should contain "**hrv**" (not case-sensitive) and should not have any return air terminals associated with them. CONTAM3DExport will perform checks on the PRJ file for these requirements. The exhaust-only and hrv simple AHSs do not have corresponding air loops associated with them in EnergyPlus. Instead, hrv systems will generate a `ZoneHVAC:EnergyRecoveryVentilator` and associated objects in the IDF file, and the exhaust-only system will generate a `Fan:ZoneExhaust` and associated objects in the IDF file.

Ducts
Ducts are currently not supported when coupling with EnergyPlus. CONTAM3DExport will not perform a check on this.

Controls
In CONTAM *Constant* (or *Set*) type controls having a name are used to receive input control signals from EnergyPlus. *Split* (or *Pass*) type controls having a name are used to send output signals to EnergyPlus from CONTAM. Prior to executing a simulation with a project that has controls, be sure to either run a CONTAM only simulation or at least a full building check via the Simulation->Run Building Check menu item to make sure that the controls are sequenced. This procedure must be followed after any new controls are added to a project, and the project must also be saved after the controls are sequenced.

An `ExternalInterface:FunctionalMockupUnitImport:From:Variable` object will be created for named *Constant* type controls, and an `ExternalInterface:FunctionalMockupUnitImport:To:Actuator` object will be created for named *Split* type controls. However, the user must set the *Name* property as well as the *Output:Variable* and *Actuated Component* properties of the `From:Variable` and `To:Actuator` objects respectively, directly within the IDF file after it has been created. The `To:Actuator` object can be replaced with either a `To:Schedule` or `To:Variable` object within the generated IDF file if desired.

Create the EnergyPlus IDF File using *CONTAM3DExport*

1. Run CONTAM3DExport and select the PRJ file to convert to EnergyPlus.
2. Select an existing IDF file that contains the `Material` and `Construction` objects that you want to incorporate into the new IDF file.
3. Select which constructions you want to associate with each surface category.
4. Select the *Building Coupled with CONTAM* Export Option.
5. Click the Save EnergyPlus IDF button. You will be prompted when the export is complete.

Files Created by *CONTAM3DExport*

- **IDF file** - contains the geometry, air loops, exhaust fans and FMU-related objects.
- **ContamFMU.fmu** - a compressed zip file that will be unpacked and used during a coupled EnergyPlus/CONTAM simulation. It contains the following files:

- **modelDescription.xml** - used by the EnergyPlus FMI co-simulation master algorithm to obtain data exchange parameters.
- **contam.prj** - the CONTAM project file used to generate the corresponding EnergyPlus IDF file and used by ContamX during the co-simulation.
- **contam.vef** - used by ContamX to obtain data exchange parameters.
- **ContamFMU.dll** - dynamic link library utilized by the EnergyPlus master algorithm to implement FMI-based program control and data exchange with ContamX.
- **contamx3.exe** - the CONTAM simulation engine controlled by the EnergyPlus FMI co-simulation master algorithm via the ContamFMU.dll.

Modify IDF File

The IDF file generated by CONTAM3DExport contains the building geometry, surface materials and constructions, air loops, exhaust fans and partial external interface objects as described above. However, the file will not include the HVAC components associated with the generation or removal of energy, e.g., heating/cooling coils, internal gains, and thermostats. The IDF file can now be edited using the *IDF Editor* or other text-based editor to add the desired HVAC system components. The IDF file can also be viewed and edited using tools such as SketchUp with the OpenStudio plug-in [Gugliemetti 2011, NREL 2015]. This is useful to modify the rectangular elements that are created by CONTAM3DExport, for example to modify roofs to have sloped surfaces and to add shading components to the building.

Run the Simulation

The IDF file and ContamFMU.fmu file must be located in the same directory. The files in the ContamFMU.fmu file can be extracted using a zip file decompression program. This may be necessary to modify run control parameters of the contam.prj file as outlined below.

1. Set the time step and the beginning and ending dates to be the same in the contam.prj file located in the ContamFMU.fmu file and the EnergyPlus IDF file.
2. Prior to running the coupled simulation, copy the contam.prj and contamx3.exe files from the ContamFMU.fmu file into another directory. Using ContamW, open the contam.prj file and set the simulation run control parameters to run a simulation for a single day that corresponds to the first day of the desired coupled simulation. Run the simulation, and it will generate a CONTAM restart file having the same name as the project file, i.e., contam.rst. Copy the RST file back to the original directory that contains the IDF and PRJ files for the co-simulation. This restart file will be used by ContamX during the warm-up period of the EnergyPlus simulations.
3. Use EPLaunch to select the IDF file and desired EnergyPlus weather (EPW) file and run the simulation.
4. Once the simulation is complete, ContamW can be used to view/plot results of contam.prj including temperatures.

6.2 ENERGYPLUS INPUT FILES

These input files are utilized for the coupling process to coordinate the transfer of data during the co-simulation between EnergyPlus and CONTAM via the ContamFMU.dll. These files can be created using the CONTAM3DExportTool, but their contents are provided here.

contam.vef

This *variable exchange file* is used by ContamFMU.dll to coordinate data exchange information at the beginning of a co-simulation run. This file will contain a list of variables that are to be exchanged during co-simulation. The number and types of variables will depend on the contents of the CONTAM project file.

In the following material individual variables are briefly described. Units are indicated within brackets, [ ]. The type of variable is in parentheses.

`(I)` = integer value,

`(R)` = real/decimal value,

`(char)` = single character

`(string)` = string of characters

File Format:

The first line is a file header indicates file type and version:

```
VariableExchangeFile 1.0
```

The second line provides the number of variables being exchanged:

```
nvariables (I)
```

Each line will provide information related to a single variable to be exchanged during co-simulation:

```
input/output indicator // (char) "I" for input to ContamX or "O" for output
  // to ContamX.
variable identification // (string) - This identifier is used to ensure the
  // variables match those of the modelDescription.xml file. The order of
  // these variables must match those in the xml file to ensure that
  // EnergyPlus (the FMI master algorithm)can match the inputs and outputs
  // with the slave process, ContamFMU.dll
variable type // (string) A three character abbreviation that indicates the
  // type of data that follows.
variable data // This information depends on the type as described below.
```

Type strings and related data (for both FMI and Type98):

- `cti` – *control input*. `CTRL_I_<name> <name>`
- `cto` – *control output*. `CTRL_O_<name> <name>`
- `zti` – *zone temperature input [C]*, `TAIR_<lev_nr>_<zone_name> lev_nr zone_name`
- `wat` – *ambient temperature [C]*. `WTH_AmbientTemp`
- `wbp` – *barometric pressure [Pa]*. `WTH_BarometricPressure`
- `wws` – *wind speed [m/s]*. `WTH_WindSpeed`

- `wws` – *wind direction [deg]*. `WTH_WindDirection`
- `mix` – *interzone airflow rate [m³/s]*. `MIX_<lev_from>_<zone_from>_<lev_to>_<zone_to> lev_from zone_from lev_to zone_to`
- `inf` – *infiltration airflow rate [m³/s]*. `INFIL_<lev_nr>_<zone_name> lev_nr zone_name`
- `ahs` – *air handling system supply air terminal [m³/s]* (input to ContamX). `AHS_supply_<lev_nr>_<zone_name> lev_nr zone_name ahs_nr number_of_zones_served`
- `ahr` – *air handling system supply air terminal* (input to ContamX). `AHS_supply_<lev_nr>_<zone_name> lev_nr zone_name ahs_nr number_of_zones_served`
- `exh` – *exhaust-only air handling system return air terminal* (input to ContamX). `AHS_exhaust_<lev_nr>_<zone_name> lev_nr zone_name ahs_nr number_of_zones_served`
- `hvs` – *energy recovery ventilator air handling system supply air terminal* (input to ContamX). `AHS_supply_<lev_nr>_<zone_name> lev_nr zone_name ahs_nr number_of_zones_served`
- `hvr` – *energy recovery ventilator air handling system return air terminal* (input to ContamX). `AHS_supply_<lev_nr>_<zone_name> lev_nr zone_name ahs_nr number_of_zones_served`
- `poa` – *air handling system percent outdoor air* (input to ContamX). `PCTOA_<AHS_name> ahs_name`

modelDescription.xml

This file is used by the EnergyPlus co-simulation master algorithm to coordinate data exchange. The general requirements of its content is defined by the FMI for co-simulation specification [MODELISAR 2010a, MODELISAR 2010b]. The unique variable names are the same as those provided for the <project>.vef, e.g., `TAIR_<lev_nr>_<zone_name>`.

6.3 GETTING HELP

This help manual is available in both printed form and accompanying the ContamW program as "on-line" help. There are several ways to access the on-line help system. You can either use the **Help Contents…** selection of the **Help** menu to display the contents of the help manual or use the context-sensitive feature to access help.

Help Contents

You can browse the contents or index of the help system to view the topic of your choice and use the search feature to display a list of help topics that contain the keyword in which you are interested.

Context-Sensitive Help

The context-sensitive help feature is designed to provide you with specific information related to the currently active ContamW window or dialog box. To activate the context-sensitive help system, press the **F1** key.

Help Index and Search

Be sure to use the on-line help index and search features if you are having trouble locating a specific topic.

Obtaining Technical Support

If you need to contact the CONTAM developers for guidance on using the program or to report a technical problem with the program you can do so via email at the addresses below. If you encounter an error while working with CONTAM, and you require assistance solving the problem, you should immediately (before running ContamW or ContamX again) make copies of the CONTAMW2.LOG, PROJECT.BKP and WEATHER.BKP located in the ContamW application data directory (C:\Documents and Settings\<User Account>\Application Data\NIST\ContamW\) and the .XLOG file located alongside the PRJ file. You can email these files to NIST so that we may be better equipped to address your particular problem.

NIST Contact Information

IAQ and Ventilation Group 301-975-6431

7 THEORETICAL BACKGROUND

This section provides theoretical background of the CONTAM program. There are basically three subsections: the first provides a summary of underlying assumptions of the model and the others address the contaminant and airflow analysis respectively.

7.1 MODEL ASSUMPTIONS

CONTAM is a powerful tool that models airflow and contaminant dispersal in buildings. It is important to realize that this tool implements mathematical relationships to model airflow and contaminant related phenomenon and therefore incorporates assumptions that simplify the model from that of the modeled phenomenon. The following is a brief description of these modeling assumptions.

Well-mixed zones – This assumption refers to the treatment of each zone as a single node, wherein the air has uniform (well-mixed) conditions throughout. These conditions include temperature, pressure and contaminant concentrations. Therefore, localized effects within a given zone cannot be accounted for using CONTAM. For example, if you utilize a contaminant source that introduces a mass of contaminant into a zone at a certain time (burst source), the contaminant will be diluted to the entire volume of the zone within a single time step.

One-Dimensional Convection/Diffusion Zones – In versions prior to CONTAM 2.3 all zones were considered to be *well-mixed*, i.e., having a uniform concentration. However, beginning with version 2.3, zones can be preconfigured by the user to be one-dimensional convection/diffusion zones in which contaminants can be allowed to vary along a user-defined axis. When operating under the newly added *short time step* method ContamX will provide for the ability of contaminant concentration gradients to occur in the direction of the convection/diffusion axis by programatically sub-dividing the zone into a series of *well-mixed* cells along the axis.

Duct Systems – Typically, during contaminant simulation, there are similarities between duct junctions and well-mixed zones and between duct segments and airflow paths. In this case the volumes of the duct junctions are determined from the duct segments to which they are connected. However, in CONTAM 2.4 the entire duct system can modeled to account for one-dimensional convection/diffusion flow through the system. This feature is available as an option under the new *short time step* contaminant simulation method.

Conservation of mass – When performing a steady-state simulation, the mass of air within each zone is conserved by the model. This implies that air can neither be created nor destroyed within a zone. However, when performing a transient simulation, CONTAM now provides the option of allowing the accumulation or reduction of mass within a zone due to the variation of zone density/pressure and the implementation of non-trace contaminants within a simulation. This is further addressed in this Theory section (see Airflow Analysis and Contaminant Analysis), as well as in a previous section addressing simulation settings (see Airflow Numerics Properties).

Trace contaminants – Trace contaminants are those that are found in low enough levels that they do not affect the density of air within a zone. You must be careful not to rely on the model to handle contaminant concentrations that would cause a change in the density of air. The program will allow for contaminants to reach levels that would, in actuality, affect the density, but the program will still treat them as if they were trace contaminants.

Non-trace contaminants – Non-trace contaminants are those that are present in such quantities that they can influence the air density, e.g., water vapor. In CONTAM you define those contaminants that are components of the air within the building. CONTAM will treat them as a mixture of ideal gases.

Thermal effects – The model does not handle heat transfer phenomenon per se, but does provide for the scheduling of zone temperatures. Zone temperatures can be either constant or allowed to change during transient simulations according to user-defined temperature schedules. CONTAM will determine airflows and non-trace contaminant mass fractions induced by temperature differences between zones including ambient (e.g., as caused by the stack effect). You can also vary the outdoor temperature for transient simulations using weather files.

Airflow paths – Airflow through various airflow elements provided by CONTAM is modeled using either a powerlaw or quadratic relationship between airflow and pressure difference across the flow path. These relationships are models themselves, and care should be taken when implementing them to represent building features within your idealized buildings. See Airflow Elements for detailed explanations of these models.

Source/sink models – CONTAM provides several different source/sink elements or representations of contaminant generation/removal processes. These elements are based upon models found throughout the literature. You should be sure to utilize models that are appropriate for the contaminant source/sink that you want to represent. See Contaminant Source/Sink Elements for detailed explanations of these models.

The previous assumptions relate to the mathematical representations utilized by CONTAM in performing analysis. These assumptions should be distinguished from assumptions made by you, the user, when creating a model of a building referred to as a building idealization (see User Tasks in the Getting Started section). Engineering judgment is required on your part to insure that your building representation is adequate for the purposes of your analysis.

7.2 Contaminant Analysis

The CONTAM contaminant dispersal model is an implementation of Axley's methods [Axley 1987 and 1988]. 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 basis for contaminant dispersal analysis is the application of conservation of mass for all species in a control volume (c.v.). A c.v. is a volume of air which may correspond to a single room, a portion of a room, or several well-coupled rooms (a CONTAM *zone*) or the ductwork (where a *junction*, under the well-mixed assumption, has half the volume of each of the adjacent duct segments). The representation of building spaces as CONTAM zones is a matter of engineering judgment.

CONTAM 2.3 added the capability to model convection-diffusion in the ductwork and user-selected zones instead of treating them as *well-mixed* control volumes. In CONTAM this is done with the new *short time step* method, which utilizes an explicit Euler formulation.

CONTAM 3.2 added a variable time step solver based on CVODE [Hindmarsh 2012], which is a general-purpose code for solving ordinary differential equations. While the remainder of this section provides details on the contaminant transport equations and original solution techniques, the CVODE solution methods are addressed in Lorenzetti [2013].

Properties of Air

In CONTAM air is treated as an ideal gas with properties computed from the ideal gas law. The density of air is given by

$$\rho = m/V = P/(RT) \tag{1}$$

where

m = the mass of air in
V = a given volume,
P = the absolute pressure,
R = the gas constant for air, and
T = the absolute temperature.

The mass of air in c.v. i is the sum of the masses of the individual contaminants, α, in the c.v.

$$m_i = \sum_\alpha m_i^\alpha \tag{2}$$

The concentration of contaminant α in c.v. i is defined as

$$C_i^\alpha = m_i^\alpha / m_i \tag{3}$$

In CONTAM *concentration* refers to a mass ratio rather than a volumetric ratio unless otherwise specified.

Air is a mixture of several different species. The value of the gas constant for the air in a c.v. is given by:

$$R_i = \sum_\alpha R^\alpha C_i^\alpha \tag{4}$$

where R_i = the gas constant of species α which equals the universal gas constant, 8 314.41 J/(kmol·K), divided by the molar mass of α (kg/kmol). Similarly, for thermal calculations (an option in CONTAM for duct flow under the *short time step* method only), the specific heat of air in a duct segment is given by the weighted sum of the specific heats of the individual species:

$$Cp = \sum_\alpha Cp^\alpha C_i^\alpha \tag{5}$$

Under typical conditions only water vapor has an impact on the properties of air and even that can be ignored as an initial approximation. There is a standard definition of species concentrations for *dry air* which yields an effective molar mass of 28.9645 kg/kmol and a gas constant of 287.055 J/(kg·K). ASHRAE often refers to dry air at *standard conditions* which are 101.325 kPa and 20 ºC and notes the density of such air is 1.20 kg/m^3 [ASHRAE 2004 p 18.4]. More precisely, the density is 1.20410 kg/m^3 as computed by equation (1).

NOTE: ASHRAE considers water vapor in terms of *humidity ratio* instead of mass concentration. The humidity ratio, *W*, is defined as the ratio of the mass of water vapor to the mass of dry air in the volume: W = m_w / m_{da} [ASHRAE 2005 p 6.8]. The CONTAM mass concentration, which ASHRAE refers to as *specific humidity*, is: $C_w = m_w / (m_w + m_{da})$. The conversions between humidity ratio and mass concentration are $C_w = W / (1+W)$ and W = $C_w / (1–C_w)$.

In many cases we are interested in species concentrations that are too small to signif icantly affect the air density (or specific heat). These are referred to as *trace concen trations*. When a simulation involves only trace contaminants, CONTAM uses dry air to compute the air properties.

7.2.1 Contaminant Concentrations

Within CONTAM a contaminant may be added to c.v. *i* by:

- inward airflows through one or more paths at the rate $\sum_j F_{j\to i}(1-\eta_j^\alpha)C_j^\alpha$ where $F_{j\to i}$ is the rate of air mass flow from c.v. *j* to c.v. *i* and η_j^α is the filter efficiency in the path
- species generation at the rate G_i^α.

A species may be removed from the c.v. by:

- outward airflows from the zone at a rate of $\sum_j F_{i\to j}C_i^\alpha$ where $F_{i\to j}$ is the rate of air mass flow from c.v. *i* to c.v. *j*, and
- species removal at the rate $R_{\alpha,i}C_{\alpha,i}$ where $R_{\alpha,i}$ is a removal coefficient.

A species may be added or removed by first-order chemical reactions with other species at the rate $\sum_{\beta} \kappa^{\alpha,\beta} m_i^{\beta}$ where $\kappa\alpha\beta$ is the kinetic reaction coefficient in c.v. *i* between species α and β. (Sign convention: positive κ for generation and negative κ for removal). This linear expression currently limits the kinds of reactions that can be modeled.

Combining these processes into a single equation for the rate of mass gain of species α in c.v. *i* gives:

$$\frac{dm_i^{\alpha}}{dt} = \sum_j F_{j\to i}(1-\eta_j^{\alpha})C_j^{\alpha} + G_i^{\alpha} + m_i \sum_{\beta} \kappa^{\alpha,\beta} C_i^{\beta} - \sum_j F_{i\to j} C_i^{\alpha} - R_i^{\alpha} C_i^{\alpha} \tag{6}$$

The transient conservation of species mass in a control volume is given by:

(mass of contaminant α in c.v. *i* at time $t+\Delta t$ **) =**

(mass contaminant α in c.v. *i* at time *t***) +**

Δt **×** **(**rate gain of contaminant α – rate loss of contaminant α**)**

Or in equation form as:

$$\rho_i V_i C_i^{\alpha}\Big|_{t+\Delta t} \approx \rho_i V_i C_i^{\alpha}\Big|_{t} + \Delta t \cdot \left[\sum_j F_{j\to i}(1-\eta_j^{\alpha})C_j^{\alpha} + G_i^{\alpha} + m_i \sum_{\beta} \kappa^{\alpha,\beta} C_i^{\beta} - \sum_j F_{i\to j} C_i^{\alpha} - R_i^{\alpha} C_i^{\alpha}\right]_{t+\delta t} \tag{7}$$

7.2.2 Numerical Calculation of Contaminant Concentrations

Several possible solutions for equation (7) can be characterized by the choice of δt to determine the rate of gain or loss. CONTAM has traditionally chosen $\delta t = \Delta t$. Equation (7) becomes:

$$\left[\rho_i V_i + \Delta t \cdot \left(\sum_j F_{i\to j} + R_i^{\alpha}\right)\right] C_i^{\alpha}\Big|_{t+\Delta t} \approx \rho_i V_i C_i^{\alpha}\Big|_{t} + \Delta t \cdot \left[\sum_j F_{j\to i}(1-\eta_j^{\alpha})C_j^{\alpha} + G_i^{\alpha} + m_i \sum_{\beta} \kappa^{\alpha,\beta} C_i^{\beta}\right]_{t+\Delta t} \tag{8}$$

All concentrations C_i^{α} at time $t+\Delta t$ are functions of various other concentrations also at $t+\Delta t$. This is the *standard implicit method*, and it requires that a full set of equations (8) must be solved simultaneously.

The number of equations, N, equals the number of species times the number of control volumes. In a traditional Gauss elimination (or LU decomposition) solution the computation time is proportional to N^3, making it impractical for large problems. CONTAM offers three solution methods which take advantage of matrix sparsity to handle cases with large numbers of equations. These are a direct skyline algorithm, an iterative biconjugate gradient (BCG) algorithm, and an iterative successive over relaxation (SOR) algorithm. (LU decomposition is provided only for testing and benchmarking.) The skyline algo rithm is very fast for problems of intermediate size but can be slow for large problems. The SOR algorithm requires much less memory and may be faster for large problems unless there are convergence difficulties. In such cases try the BCG solution, although it may also experience convergence difficulties. It can be useful to test the different methods to determine which will give optimum performance before doing a long transient simulation.

A more accurate solution can be obtained by choosing $\Delta t = \Delta t/2$ which means average conditions during the time step. This has been implemented in CONTAM by a trapezoidal integration which still requires solving the full set of simultaneous equations.

We can also choose $\Delta t = 0$. In this case equation (7) becomes:

$$\rho_i V_i C_i^\alpha \Big|_{t+\Delta t} \approx \rho_i V_i C_i^\alpha \Big|_t + \Delta t \cdot \left[\sum_j F_{j\to i}(1-\eta_j^\alpha) C_j^\alpha + G_i^\alpha + m_i \sum_\beta \kappa^{\alpha,\beta} C_i^\beta - \sum_j F_{i\to j} C_i^\alpha - R_i^\alpha C_i^\alpha \right]_t \qquad (9)$$

Every concentration C_i^α at time $t+\Delta t$ is a function of various other known concentrations at time t. This is the *standard explicit method* which has the computational advantage of not requiring the solution of simultaneous equations. That advantage is offset by instability under some conditions. That is, the concentrations at successive time steps may diverge wildly from the analytically correct solution. Stability is determined by the magnitudes of the coefficients in equations (7). For example, when the sum of the flows into or out of the c.v. in one time step is greater than the mass of air in the c.v., the solution becomes unstable, that is, values at successive time steps begin to oscillate around the true solution and eventually reach impossible values.

Instability will also result when $\kappa^{\alpha,\beta} \cdot \Delta t$ or $R_i^\alpha \cdot \Delta t$ exceed a certain magnitude. The standard implicit method is stable at all time steps.

The stability question is so important that it is useful to review the time scales that can be expected in the normal operation of buildings. ASHRAE indicates that the air exchange rate to condition and ventilate rooms should be less than 12 air changes per hour in nearly all commercial applications. [ASHRAE 1999b, p. 3.2 Table 1 - General Design Criteria] This corresponds to a stability limit for the explicit method of 1/12 h, or 5 min. In CONTAM the volume of a junction is half the sum of the volumes of the duct segments meeting at the junction. A junction between two 2 m long duct segments where the air is flowing at a velocity of 2 m/s would have a stability limit of 1 second. Smaller limits are likely to occur because of shorter duct segments or higher velocities.

In early versions of CONTAM it was anticipated that the processes being modeled would require time steps down to about 5 minutes. The very short stability limit for modeling the ductwork drove CONTAM to use an implicit solution where the execution time for relatively few time steps was less than doing many shorter times steps using the explicit solution. The recent addition of control system modeling and the need to track quick contaminant releases both require time steps no longer than a few seconds. We will therefore reconsider use of the explicit model for the ducts.

The following figure shows a CONTAM sketchpad representation of the typical features of a very simple building and its air handling system. The duct icons indicate the normal direction of flow. The ductwork consists of a return duct with terminals (R) in each room, an exhaust to ambient (X), a path for recirculation, an outdoor air intake (OA), and a supply duct with terminals (S) in each room.

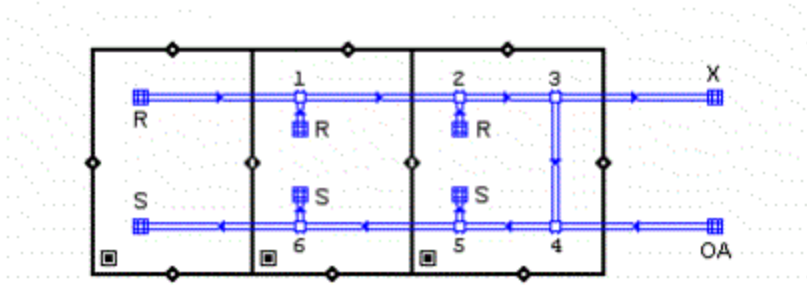


Figure – Schematic of a basic air handling system

When the simulation time step is sufficiently short, the *explicit method* can be used to compute the contaminant concentrations in the zones. That same time step can be used with the *implicit method* to compute the concentrations in the duct junctions if the calculations are performed in the proper sequence where equation (8) is solved one node at a time following the direction of airflow. The following discussion will show how the duct junction concentrations can be solved implicitly, thereby avoiding stability problems, and without solving simultaneous equations, thereby producing a fast computation.

The process begins by computing all airflows based on conditions known at time *t*. The contaminant calculation sequence then begins by using a modification of equation (9) to compute the concentrations C_i^{α} at time $t+\Delta t$ in all zones:

$$(\rho_i V_i + \Delta t \cdot \sum_d F_{i\to d})\, C_i^{\alpha}\Big|_{t+\Delta t} \approx \rho_i V_i\, C_i^{\alpha}\Big|_t + \Delta t \cdot \left[\sum_j F_{j\to i}(1-\eta_j^{\alpha})C_j^{\alpha} + G_i^{\alpha} + m_i \sum_{\beta} k^{\alpha\beta} C_i^{\beta} - \sum_j F_{i\to j} C_i^{\alpha} - R_i^{\alpha} C_i^{\alpha}\right]_t \qquad (10)$$

In equation (10) the flows out of the zone are divided into two parts

$\sum_j F_{i\to j}$, flows to other zones, and

$\sum_d F_{i\to d}$, flows into duct junctions through return terminals or leaks,

to conserve contaminant mass when computing concentrations at the return terminals.

The concentrations in the return terminals (R and OA) are then computed with a simplified form of equation (8), because sources, sinks, and reactions are not modeled in ducts, although these features could be added in the future.

$$\left[\rho_i V_i + \sum_j F_{i\to j} + R_i^{\alpha}\right] C_i^{\alpha}\Big|_{t+\Delta t} \approx \rho_i V_i\, C_i^{\alpha}\Big|_t + \Delta t \cdot \left[\sum_j F_{j\to i}(1-\eta_j^{\alpha})C_j^{\alpha}\right]_{t+\Delta t} \qquad (11)$$

This calculation is particularly simple for terminals because there is only one airflow into each return terminal and an equal airflow out into the duct network. The concentration in the flow into the terminal, C_j^{α} at time $t+\Delta t$, comes from equation (10) for the surrounding zone, *j*. Once all return terminal concentrations have been computed the upstream concentrations are known for both ducts meeting at the left-most junction (# 1) in the return duct, and equation (11) can be solved directly because all concentrations on the right side of the equation are known. Once the

junction #1 concentrations have been computed all upstream concentrations are known for the next junction to the right (# 2) along the return duct and its concentrations can then be computed. This process continues around the ductwork until the concentrations at all junctions and all supply terminals have been computed. This implicit calculation of the concentrations in the duct junctions is unconditionally stable, so overall stability is determined by conditions in the zones.

CONTAM 2.4 adds this process calling it the *short time step* method (STS).

Reactions may include coefficients that lead to instabilities for an explicit simulation. Very fast reactions would produce such coefficients. Therefore, the STS method processes reactions by an implicit calculation involving only the contaminants in the c.v. after all other calculations have been performed for the time step.

7.2.3 One-Dimensional Convection-Diffusion Flow

Although it is possible to use a time step as short as one second in CONTAM, this may not produce a more accurate simulation of transient concentrations because of the use of control volumes which are treated as well-mixed. For example, consider the sudden release of a contaminant at one end of a 10 m long zone with air flowing at a typical 0.2 m/s from the release point toward a doorway at the other end. It would take about 50 seconds for the contaminant to reach that doorway and begin entering the next zone. In the well-mixed zone model the sudden release will instantly produce a uniform concentration (equal to the mass released divided by the mass of air) everywhere in the zone, even at the far end 10 m from the source, and begins entering the adjacent zone within one time step.

The well-mixed zone model is appropriate when the time step is longer than the mixing time of the zone. Conventional HVAC systems attempt to produce well mixed zones, but the mixing time is on the order of a few minutes rather than seconds.

The standard numerical solution to this problem is to create smaller control volumes whose size is similar to the distance traveled in one time step. This is done in the computational fluid dynamics (CFD) models that consider mass, momentum, and energy in computing the flow field in a space that has been divided into *many* control volumes. Conventional CFD might divide a volume into 30 cells along each of the three directions. This increases the computation effort from one large "cell" per zone to 27,000 (=30×30×30) cells per zone. This analysis approach is not practical for a building containing many zones.

One-dimensional convection-diffusion flow has been introduced to CONTAM as a compromise between simple, and fast, well-mixed zones and a full CFD simulation. CONTAM 2.4 includes the option of modeling detailed contaminant migration in one, user-defined direction through a zone and or an entire duct system. This one-dimensional model is obviously appropriate for flow through a duct and reasonably appropriate for a long hallway or a zone using a displacement ventilation system. Its use in more conventional well-mixed zones is problematic because of the presence of supply air jets and areas of recirculation.

Contaminant flow in one direction consists of a mixture of *convection*, the bulk movement of air, and *diffusion*, the mixing of the contaminant within the air. CONTAM's primary 1-D convection diffusion model is taken directly from the finite volume method developed by Patankar [Patankar 1980] and described in more detail by Versteeg and Malalasekera [Versteeg and Malalasekera 1995]. This model divides the zone into a number of equal-length cells and uses an implicit method (with a fast tri-diagonal equation solver) to guarantee stability in computing the

contaminant concentrations. It has been observed that the accuracy of this method declines as the ratio of the flow velocity × time step to the length of the cell increases.

This loss of accuracy is a particular problem in the ducts where flow velocities can be quite high. CONTAM uses a Lagrangian model to handle high speed flows in ducts. In the Lagrangian model, air flowing at velocity u will create a cell $u\Delta t$ long at the inlet end of the duct segment and cause the cell at x_j to move to $x_j + u\Delta t$ during a time step of Δt. The length of the cell, Δx_j, is unchanged. This process of adding cells at the inlet end of the duct and deleting cells at the outlet end handles convection exactly. During that time step the contaminant will diffuse between adjacent cells due to molecular diffusion and turbulent mixing. That diffusion is solved by a standard implicit method using a tri-diagonal equation solver.

The cell at the outlet end of the duct segment will not necessarily have an edge at $x = L$ in which case an interpolation is necessary to compute the concentration at $x = L$, which becomes the input concentration to the next duct segment downstream. When two or more duct segments merge at a junction the contaminant concentration at the junction is the flow-weighted average of the concentrations at the end of each incoming duct.

High velocities produce long cells and, therefore, relatively few equations to be solved in a duct segment. At lower velocities more cells are required, and as the velocity approaches zero the number of cells approaches infinity. To prevent this, ContamX automatically switches to the Eulerian finite volume model when $u\Delta t$ is less than the user specified minimum cell length. This corresponds to the flow regime where the finite volume is most accurate.

The axial dispersion coefficient in ducts is computed from the following relations. For laminar flow (Re < 2000) the Taylor-Aris relation is used [Wen & Fan 1975, p. 127]:

$$E = D_m + \frac{\bar{u}^2 d^2}{192 D_m}; \quad \frac{L}{d} < 0.04 \frac{\bar{u} d}{D_m} \tag{12}$$

where

E = axial dispersion coefficient [m²/s],
D_m = molecular diffusion coefficient [m²/s],
$\bar{u}$ = average fluid velocity [m/s],
d = duct diameter [m], and
L = length of duct [m].

The condition refers to a minimum length of duct for full development of laminar flow. Until another relation is found for undeveloped flow, equation (12) will be used for all laminar flows.

For turbulent flow E depends only on the Reynolds number, Re [Wen and Fan 1975, p. 149]:

$$\frac{E}{\bar{u} d} = \frac{3.0 \times 10^7}{\mathrm{Re}^{2.1}} + \frac{1.35}{\mathrm{Re}^{0.125}} \tag{13}$$

Duct Thermal Model

When performing simulations using the *short time step* method, a simple duct thermal model has been incorporated into the solution that may be implemented as an option to the user. This model addresses only convective heat transfer.

Conservation of thermal energy is analogous to the conservation of contaminant mass in a control volume:

(energy at time $t+\Delta t$) = (energy at time t) + Δt •(rate of energy gain – rate of energy loss)

Replace the concentration terms, C_i^{α}, in equation (8) with specific energy, Cp_iT_i:

$$\left(\rho_i V_i + \Delta t \cdot \sum_j F_{i\to j} \right) Cp_i T_i\big|_{t+\Delta t} \approx \rho_i V_i Cp_i T_i\big|_t + \Delta t \cdot \sum_j F_{j\to i} Cp_j T_j\big|_{t+\Delta t} \tag{14}$$

where

Cp_i = specific heat of c.v. i, and

T_i = temperature of c.v. i.

This is done only for ducts because the energy in the moving air dominants the solution in a well designed and well built duct system. That is not the case for zones where conductive and airflow heat transfers are of a similar scale. This model ignores heat exchange with the ductwork and its transient impact. However, the model should still be a useful help in computing the draft of a chimney.

7.2.4 Contaminant Source/Sink Elements

CONTAM allows you to define the generation, G_α, and removal, R_α, coefficients for some simple cases.

Constant Coefficient Model

The constant coefficient or general source/sink model uses the following equation:

$$S_\alpha(t) = G_\alpha - R_\alpha C_\alpha(t) \tag{15}$$

where S_α is called the contaminant α "source strength". The CONTAM internal units for the terms in equation (16) are: C_α [kg_α / kg_{air}], G_α, S_α [kg_α / s], and R_α [kg_{air} / s]. You 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 $R = f \bullet e$ 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.

Pressure Driven Model

The pressure driven 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:

$$S_\alpha = G_\alpha (P_{amb} - P_i)^n \tag{16}$$

Cutoff Concentration Model

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

$$S_\alpha(t) = G_\alpha \left(1 - \frac{C_\alpha(t)}{C_{cutoff}} \right) \tag{17}$$

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

Decaying Source Model

Another source equation for volatile organic compounds is the exponentially decaying source which is expressed in the form

$$S_\alpha(t) = G_\alpha e^{-t/t_c} \tag{18}$$

where

S_α = the contaminant source strength,
G_α = the initial emission rate,
t = the time since the start of emission, and
t_c = the decay time constant.

Boundary Layer Diffusion Controlled Model

The boundary layer diffusion controlled reversible source/sink model follows the descriptions presented by Axley 1991. The rate at which a contaminant is transferred into the sink is

$$h \cdot \rho \cdot A\left(C_i - \frac{C_s}{k}\right) \tag{19}$$

where

h = average film mass transfer coefficient over the sink,
ρ = 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, and
k = Henry adsorption constant or partition coefficient.

Burst Source Model

A user-specified mass of contaminant is added to a zone in a single time step, effectively an instantaneous addition – nothing can be resolved at less than one time step.

Deposition Velocity Sink Model

The deposition velocity model provides for the input of a sink's characteristic in the familiar term of deposition velocity. The deposition velocity model equation is:

$$R_\alpha(t) = v_d A_s m \rho_{air}(t) C_\alpha(t) s(t) \tag{20}$$

where

$R_\alpha(t)$ = removal rate at time t
v_d = deposition velocity
A_s = deposition surface area
m = element multiplier
$\rho_{air}(t)$ = density of air in the source zone at time t
$C_\alpha(t)$ = concentration of contaminant α at time t [M_α / M_{air}]
$s(t)$ = schedule or control signal value at time t [-]

Deposition Rate Sink Model

The deposition rate model provides for the input of a sink's characteristic in the familiar term of deposition or removal rate. The deposition rate model equation is:

$$R_{\alpha}(t) = k_d V_z \rho_{air}(t) C_{\alpha}(t) s(t) m \tag{21}$$

where

k_d = Deposition rate [1/T]

V_z = Zone volume [M^3]

other terms are the same as for the Deposition Velocity Sink Model.

7.3 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 report 50 different computer programs for multizone airflow analysis [Feustal and Dieris 1992]. Note that "zones" go by many other names in these programs, e.g., nodes, cells, and rooms are common alternatives. The airflow calculations in CONTAM are based on the algorithms developed in AIRNET [Walton 1989a and 1989b].

7.3.1 Basic Equations

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

$$F_{j,i} = f\left(P_j - P_i\right) \tag{1}$$

The mass of air, m_i [kg], in zone i is given by the ideal gas law

$$m_i = \rho_i V_i = \frac{P_i V_i}{R T_i} \tag{2}$$

where

V_i = zone volume [m^3],
P_i= zone pressure [Pa],
T_i= zone temperature [K], and
R= 287.055 [J/kg·K] (gas constant for air).

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

$$\frac{\partial m_i}{\partial t} = \rho_i \frac{\partial V_i}{\partial t} + V_i \frac{\partial \rho_i}{\partial t} = \sum_j F_{j,i} + F_i \tag{3}$$

$$\frac{\partial m_i}{\partial t} \approx \frac{1}{\Delta t}\left[\left(\frac{P_i V_i}{R T_i}\right)_t - \left(m_i\right)_{t-\Delta t}\right] \tag{4}$$

where

m_i= mass of air in zone i,

$F_{j,i}$= 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.

CONTAM 1.0 did not provide for such non-flow processes and flows were evaluated by assuming quasi-steady conditions leading to the following equation

$$\sum_j F_{j,i} = 0 \tag{5}$$

CONTAM can now provide for such non-flow processes by allowing the density to vary during time steps when performing transient simulations.

You can activate this option with the *Vary Density During Time Step* setting under the Airflow Numerics Simulation Parameters. If this parameter is set then, equation 3 is implemented when performing airflow calculations; otherwise equation 5 is used.

7.3.2 Solving the Equations

The steady-state airflow analysis for multiple zones requires the simultaneous solution of equation (5) for all zones. Since the function in equation (1) 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 and 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, $\{\mathbf{P}\}^*$, is computed from the current estimate of pressures, $\{\mathbf{P}\}$, by

$$\{\mathbf{P}\}^* = \{\mathbf{P}\} - \{\mathbf{C}\} \tag{6}$$

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

$$[\mathbf{J}]\{\mathbf{C}\} = \{\mathbf{B}\} \tag{7}$$

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

$$\mathbf{B}_i = \sum_j F_{j,i} \tag{8}$$

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

$$\mathbf{J}_{i,j} = \sum_i \frac{\partial F_{j,i}}{\partial P_j} \tag{9}$$

In equations (8) and (9) $F_{j,i}$ and $\partial F_{j,i}/\partial P_j$ are evaluated using the current estimate of pressure $\{\mathbf{P}\}$. The ContamX 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 (7) 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 $[\mathbf{J}]$ 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 presented in [Dhatt 1984] was chosen. This method can be used to solve equations with symmetric or asymmetric 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. CONTAM provides two solution methods for the linear equations: Skyline (also called profile method) and Pre-conditioned Conjugate Gradient (PCG). PCG may be useful for problems with many zones and junctions.

Analysis of the element models will show that

$$\left|\mathbf{J}_{i,i}\right| = \sum_{j \neq i} \left|\mathbf{J}_{i,j}\right| \tag{10}$$

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 had no unique solution. The ContamW user interface insures the correct interconnection of 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 (7) 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 outdoor) air is treated as a constant pressure zone. The ambient zone pressure is assumed to be zero for the flow calculation 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, when equation (4) is satisfied for all zones for the current system pressure estimate, the solution has converged. Sufficient accuracy is attained by testing for relative convergence at each zone:

$$\frac{\left|\sum_j F_{j,i}\right|}{\sum_j \left|F_{j,i}\right|} < \varepsilon \tag{11}$$

with a test ($\Sigma|F_{j,i}| < \varepsilon 1$, the absolute convergence factor) 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 ($\varepsilon = 0$).

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 [Wray 1993] indicate that the use of a simpler constant under-relaxation coefficient produces a faster, reliable convergence acceleration process. Equation (6) for the iteration process becomes

$$\{\mathbf{P}\}^* = \{\mathbf{P}\} - \omega\{\mathbf{C}\} \tag{12}$$

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:

$$\gamma = \frac{\sum_i \left| \sum_j F_{j,i} \right|}{\sum_i \sum_j \left| F_{j,i} \right|} \tag{13}$$

When $\gamma^* < \alpha\gamma$, ω is set to 1. Currently CONTAM uses α = 30%. This often reduces the number of iterations. This is simple under-relaxation. CONTAM also may alternatively use a simple trust region method implemented by David M. Lorenzetti based on [Dennis and Schnabel 1996].

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:

$$F_{j,i} = c_{j,i} + b_{j,i}\left(P_j - P_i\right) \tag{14}$$

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

$$[\mathbf{A}]\{\mathbf{P}\} = \{\mathbf{B}\} \tag{15}$$

Matrix [**A**] in equation (15) has the same sparsity pattern as [**J**] in equation (7) 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 used by CONTAM. 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.

7.3.3 Airflow Elements

Infiltration is the result of air flowing through openings, large and small, intentional and accidental, in the building envelope. Simulation programs require a mathematical model of the flow characteristics of the openings. For a general introduction see Chapter 27 of [ASHRAE 2005] and section 2.2 of [Feustel 1990].

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

$$\Delta P = \left(P_1 + \frac{\rho V_1^2}{2}\right) - \left(P_2 + \frac{\rho V_2^2}{2}\right) + \rho g\left(z_1 - z_2\right) \tag{16}$$

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/s^2)
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

$$\Delta P = P_j - P_i + P_S + P_W \tag{17}$$

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 (17) 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 (9) are related by $\partial w/\partial P_j = -\partial w/\partial P_i$ which establishes the relation in equation (10). Many forms of airflow elements are available in CONTAM.

Powerlaw Flow Elements

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:

$$Q = C(\Delta P)^n \tag{18}$$

The volumetric flow rate, Q [m^3/s], is a simple function of the pressure drop, ΔP [Pa], across the opening. A common variation of the powerlaw equation is:

$$F = C(\Delta P)^n \tag{19}$$

where the mass flow rate, F [kg/s], is a simple function of the pressure drop. A third variation is related to the orifice equation:

$$Q = C_d A \sqrt{\frac{2\Delta P}{\rho}} \tag{20}$$

where

C_d= discharge coefficient, and
A= orifice opening area.

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 primary advantage of equations (18-20) for describing airflow components is the simple calculation of the partial derivatives for the Newton's method solution of the simultaneous equations:

$$\frac{\partial F_{ji}}{\partial P_j} = \frac{nF_{ji}}{\Delta P} \quad \text{and} \quad \frac{\partial F_{ji}}{\partial P_i} = \frac{-nF_{ji}}{\Delta P} \tag{21}$$

The sign in equations (21) will agree with the sign of F. However, there is also a problem with equations (21): 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. Equations (18-20) can be replaced by

$$F = \frac{C_k \rho \Delta P}{\mu} \tag{22}$$

where

C_k= laminar flow coefficient, and
μ= viscosity.

The partial derivatives are simple constants:

$$\frac{\partial F_{ji}}{\partial P_j} = \frac{C_k P}{\mu} \quad \text{and} \quad \frac{\partial F_{ji}}{\partial P_i} = \frac{-C_k \rho}{\mu} \tag{23}$$

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] and [Isaacs 1980]. Although there is physical reason for using equation (22) at low pressure drops, its purpose here is to assure convergence of the equations when Δ*P* 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. Because the linear flow expression is not used as a true flow model but as a mathematical artifice, it is not necessary to adjust its flow coefficient. Given the uncertainty in estimating the temperature of the air as it flows through an opening, especially a crack, this additional detail is of debatable usefulness.

The CONTAM functions for powerlaw elements calculate flows using both the laminar and the turbulent models and select the method giving the smaller magnitude flow. There is a discontinuity in the derivative of the $F(\Delta P)$ curve where the two equations intersect. This discontinuity is a violation of one of the sufficient conditions for convergence of Newton's method [Conte and 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.

Temperature Dependence

It is useful to think of the coefficient *C* as a simple constant, C_α, evaluated at a particular set of conditions (μ_0, ρ_0 and $\nu_0=\mu_0/\rho_0$) multiplied by a correction factor to account for actual air properties. Equations (18-20) are converted to a common form and summarized below with their appropriate temperature correction factors.

$\Delta P > 0$	$\Delta P < 0$	Correction Factor
$F_{ji} = K_a C_a \rho_j (\Delta P)^n$	$F_{ji} = -K_a C_a \rho_i (-\Delta P)^n$	$K_a = (\rho_o / \rho)^n (\nu_o / \nu)^{2n-1}$
$F_{ji} = K_b C_b \sqrt{\rho_j} (\Delta P)^n$	$F_{ji} = -K_b C_b \sqrt{\rho_i} (-\Delta P)^n$	$K_b = (\rho_0 / \rho)^{n-1/2} (\nu_o / \nu)^{2n-1}$
$F_{ji} = K_c C_c (\Delta P)^n$	$F_{ji} = -K_c C_c (-\Delta P)^n$	$K_c = (\rho_o / \rho)^{n-1} (\nu_o / \nu)^{2n-1}$

(24)

CONTAM uses the following formulae for computing ρ and ν:

$\rho = P / (287.055\ T)$

$\mu = 3.7143\times10^{-6} + 4.9286\times10^{-8}T$

$\nu = \mu / \rho$

Using reference conditions of standard atmospheric pressure and 20°C gives $\rho_o = 1.2041$ kg/m^3 and $\nu_o = 1.5083\times10^{-5}$ m^2/s.

Fitting Powerlaw Coefficients

Experimental data can be used to determine the coefficients in the orifice form of the powerlaw equation:

$$F = \pm C_b \sqrt{\rho_0}\left(\pm \Delta P\right)^n \qquad \left(K_b = 1\right) \tag{25}$$

If n is known or can be assumed, C_b, in equation (24), can be computed from the inverse of equation (25)

$$C_b = \frac{F}{\sqrt{\rho_0}\left(\Delta P\right)^n} \tag{26}$$

When two points (F_1, ΔP_1) and (F_2, ΔP_2), are known, n can be computed from:

$$n = \frac{\ln\left(F_1\right) - \ln\left(F_2\right)}{\ln\left(\Delta P_1\right) - \ln\left(\Delta P_2\right)} \tag{27}$$

with C_b then computed from equation (26).

Leakage Areas

The powerlaw model can be used with the component leakage area formulation which has been used to characterize openings for infiltration calculations [ASHRAE 2001, p. 25.18]. The leakage area is based on a series of pressurization tests where the airflow rate is measured at a series of pressure differences ranging from about 10 Pa to 75 Pa. The effective leakage area is based on a rearrangement of equation (20)

$$L = \frac{Q_r \sqrt{\rho / 2\Delta P_r}}{C_d} \tag{28}$$

where

L= equivalent or effective leakage area [m^2],
ΔP_r= reference pressure difference [Pa],
Q_r= predicted airflow rate at ΔP_r (from curve fit to pressurization test data) [m^3/s], and
C_d= discharge coefficient.

There are two common sets of reference conditions:

$C_d = 1.0$ and $\Delta P_r = 4$ Pa

or

$C_d = 0.6$ and $\Delta P_r = 10$ Pa.

A leakage area can be converted to the flow coefficient by

$$C_b = LC_d \sqrt{2}(\Delta P_r)^{1/2-n} \tag{29}$$

This equation requires a value for *n*. If it is not reported with the test results, a value between 0.6 and 0.7 is reasonable.

Stairwells

A stairwell will normally be modeled as a vertical series of zones connected by low resistance openings through the floors. The CONTAM model for airflow in stairwells is based on a fit to experimental data [Achakji and Tamura 1988]. They expressed the airflow resistance per floor as an effective area A_e in the orifice equation (20) with a 0.6 discharge coefficient. The effective area is expressed in terms of the area of the shaft A_S, the distance between floors *h*, the density of people on the stairs *d*, and whether the treads are open or closed. 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/m^2. For open treads the effective area is approximately

$$A_e \approx A_s(0.089h)\left(1.0 - 0.14\sqrt{d}\right) \tag{30}$$

and for closed treads

$$A_e \approx A_s(0.083h)\left(1.0 - 0.24\sqrt{d}\right) \tag{31}$$

The coefficients for the powerlaw equation are

$$n = 0.5 \quad \text{and} \quad C_b = 0.6 \cdot \sqrt{2} \cdot A_e \tag{32}$$

Cracks

A relationship for flow through cracks that can be converted directly into a powerlaw airflow element is presented in [Clarke 1985, p. 204] as:

$$Q = ka(\Delta P)^n \tag{33}$$

where

$$n = 0.5 + 0.5\exp(-W/2) \tag{34}$$

and

$$k = 0.0097 \cdot (0.0092)^n \tag{35}$$

with

W = crack width (mm), and

a = crack length (m).

Therefore, the coefficients in the powerlaw equation (19) are given by *n* in equation (34) and

$$C_b = \sqrt{\rho_0}\ \ a\ \ 0.0097\left(0.0092\right)^n \tag{36}$$

Quadratic Flow Elements

Baker, Sharples, and Ward [Baker 1987] indicate that infiltration openings can be more accurately modeled by a quadratic relationship of the form

$$\begin{aligned} Q, \Delta P > 0 \qquad & \qquad Q, \Delta P < 0 \\ \Delta P = AQ + BQ^2 \qquad & \qquad \Delta P = AQ - BQ^2 \end{aligned} \tag{37}$$

This form can be used as an airflow element by solving the quadratic equation for F (= ρQ). Letting $a = A/\rho$ and $b = B/\rho^2$ allows equations (37) to be rewritten as

$$\Delta P = aF + bF^2 \qquad and \qquad \Delta P = aF - bF^2 \tag{38}$$

These quadratic equations solve as

$$F = \frac{\sqrt{a^2 + 4b\Delta P} - a}{2b} \qquad and \qquad F = \frac{a - \sqrt{a^2 - 4b\Delta P}}{2b} \tag{39}$$

with the partial derivatives given by

$$\frac{\partial F_{j,i}}{\partial P_i} = \frac{1}{a + 2b\left|F_{j,i}\right|} \qquad and \qquad \frac{\partial F_{j,i}}{\partial P_i} = \frac{-1}{a + 2b\left|F_{j,i}\right|} \tag{40}$$

Equations (39) require that b be nonzero to prevent a division by zero and equations (40) requires that a be nonzero to prevent a division by zero as F goes to zero. There are contrary opinions that the powerlaw relationship is better.

Temperature Dependence

It is useful to think of the coefficients a and b as simple constants evaluated at a particular set of conditions (μ_0, ρ_0 and $\nu_0=\mu_0/\rho_0$) multiplied by correction factors to account for actual air properties as was done for the powerlaw equations. That is

$$A = \left(\frac{\nu}{\nu_0}\right)A_0 \qquad and \qquad a = \left(\frac{\nu}{\nu_0}\right)a_0 \tag{41}$$

and

$$B = \left(\frac{\rho_0}{\rho}\right)B \qquad and \qquad b = \left(\frac{\rho_0}{\rho}\right)b_0 \tag{42}$$

Note that $\rho_0/\rho = T/T_0$ for a perfect gas of constant composition.

Fitting Quadratic Coefficients

The quadratic coefficients can also be computed from measured flow and pressure data. Given two points (F_1, ΔP_1) and (F_2, ΔP_2), the values of a_0 and b_0 are:

$$b_0 = \frac{\left(\frac{\Delta P_1}{F_1} - \frac{\Delta P_2}{F_2}\right)}{F_1 - F_2} \tag{43}$$

and

$$a_0 = \frac{\Delta P_1}{F_1} - b_0 F_1 \tag{44}$$

The main advantage of the quadratic model over the powerlaw model is computation speed achieved by avoiding the slow power function. (Tests have shown pow(x) four to eight times slower than sqrt(x). This performance is hardware and software dependent.) It must still be determined which model is the most accurate representation for a particular airflow element. For example, it has been found that the powerlaw model is a better approximation for smooth ducts while the quadratic model is a better approximation for rough ducts. However, for very large openings the derivatives (40) can become quite large leading to slow convergence of the simultaneous mass balance equations.

Crack Description

Theoretical relationships have been developed between the coefficients A and B of the quadratic airflow element model and the physical characteristics of the openings [Baker 1987]. These are

$$A_0 = 12\mu_0 z / Ld^3 \qquad and \qquad B_0 = \rho_0 C / 2d^2 L^2 \tag{45}$$

where

- μ= viscosity,
- ρ= density,
- *z*= distance along the direction of flow,
- *d*= crack width,
- *L*= crack length, and
- *C*= 1.5 + number of bends in the flow path.

For the mass flow form of the equations (38) the coefficients are

$$a_0 = 12\mu_0 z / \rho_0 Ld^3 \qquad and \qquad b_0 = C / 2\rho_0 d^2 L^2 \tag{46}$$

Ducts

The theory of flows in ducts (and pipes) is well established and summarized in Chapter 35 of the *2005 ASHRAE Fundamentals Handbook* [ASHRAE 2005] and treated more extensively in [Blevins 1984] in a chapter on pipe and duct flow. See the *Duct Fitting Database* [ASHRAE 2002] for extensive data on dynamic losses in duct fittings. Analysis is based on Bernoulli's equation and its assumptions. The friction losses in a section of duct or pipe are given by

$$\frac{\partial m_i}{\partial t} = \rho_i \frac{\partial V_i}{\partial t} + V_i \frac{\partial \rho_i}{\partial t} = \sum_j F_{j,i} + F_i \tag{47}$$

where

f = friction factor,
L = duct length, and
D = hydraulic diameter.

The dynamic losses due to fittings and so forth are given by

$$\Delta P_d = C_d \frac{\rho V^2}{2} \tag{48}$$

where C_d = dynamic loss coefficient. Total pressure losses are given by

$$\Delta P = \Delta P_f + \sum \Delta P_d \tag{49}$$

Since $F = \rho VA$, where A is the cross section (or flow) area,

$$F = \sqrt{\frac{2\rho A^2 \Delta P}{fL/D + \sum C_d}} \tag{50}$$

CONTAM calculates the friction factor using the nonlinear Colebrook equation [ASHRAE 2001, p 2.9, eqn. 29b]:

$$\frac{1}{\sqrt{f}} = 1.44 + 2 \cdot \log(D/\varepsilon) - 2 \cdot \log\left(1 + \frac{9.3}{\mathrm{Re} \cdot \varepsilon / D \cdot \sqrt{f}}\right) \tag{51}$$

where

ε = roughness dimension, and
Re = Reynolds number = $\rho VD/\mu = FD/\mu A$.

This nonlinear equation may be readily solved using the following iterative expression derived from equation (51) by Newton's method:

$$g^* = g - \frac{g - \alpha + \gamma \ln(1 + g\beta)}{1 + \gamma\beta/(1 + g\beta)} \tag{52}$$

where

$g = f^{-1/2}$,
$\alpha = 1.14 - \gamma \ln(\varepsilon/D)$,
ß = $9.3/(\mathrm{Re} \cdot \varepsilon/D)$, and
$\gamma = 2 \cdot \log(e) = 0.868589$.

The convergent solution is achieved in 2 or 3 iterations of equation (52) using $g = \alpha$ as a starting value. If the value of g has been saved from the previous time it was computed for a particular duct element, and the flow rate has not changed greatly, only one iteration of equation (52) will be needed to compute the friction factor.

The exact derivatives of equation (50) are difficult to compute, so CONTAM uses a secant approximation. The derivatives suffer the standard problem of powerlaw equations, i.e., they go undefined as ΔP approaches zero. This is solved in CONTAM by the linear approximation (22) with the coefficient computed to give the same flow as equation (50) at the user specified

transition Reynolds number (default = 2000). A more detailed description of the flow in the laminar region could be developed, but that would probably exceed the level of detail with which the rest of the problem is described in CONTAM.

Terminal Loss Calculation

In CONTAM, duct terminals are essentially zero-length ducts characterized by an equivalent loss coefficient that is determined during simulation from the properties of the terminal, C_t and C_b, and the schedule or control signal (S) acting upon the terminal.

$C_e = C_t + C_b + (1\text{-}S)/S$

where

C_t = terminal loss coefficient

C_b= terminal balance coefficient

S_c = schedule or control signal acting upon the terminal

$S = \max(1\text{x}10^{-6}, S_{c1})$

$S_{c1}= \min(1, S_c)$

Using this technique, an input signal S_c of 0.0 will effect a very high loss coefficient, that of 1.0 will simply provide the loss determined by the combination of C_t and C_b, $S_c$between $1\text{x}10^{-6}$ and 1 will be used to increase the value of ($C_t + C_b$) according to the equation above, and S_c less than 0.0 will equate to 1.0, i.e., the loss can not be reduced below ($C_t + C_b$). The value C_e will then be used as the dynamic loss coefficient in the duct flow equations presented above.

Fans

This section describes the theory of how forced flow elements are modeled with CONTAM. Forced flow elements include constant flow fans and variable flow fans. Constant flow fans include both constant volume and constant mass flow. Variable flow fans are modeled based on the input of a fan performance curve that relates pressure drop to airflow through the fan flow element.

Constant Flow Fans

One particularly simple but useful airflow element sets a constant flow between two nodes. Since the flow is constant, the partial derivatives of flow with respect to the node pressures must be zero. The constant flow element does not contribute to the Jacobian, [**A**], but it does add to the right side vector, {**B**}.

Constant flow elements do not mathematically link the pressures of the adjacent nodes. It is necessary that all node in the network be linked to constant pressure nodes in order to have a unique solution. Violation of this restriction will produce a division by zero somewhere in the solution of the equations. Consider the following simple network:

```
|----C1----|----F1----|----C2----|----F2----|----C3----|
P1         P2         P3         P4         P5         P6
```

where P_1 and P_6 are known pressures, and F_1 and F_2 are known flows. Since the flow through C_2 is determined by P_3 - P_4, and no other flow is related to those pressures, there are not enough

equations to determine P_3 *and* P_4 uniquely. This is a mathematical expression of the fact that it is possible for F_1 and F_2 to be assigned different flows, which produces a physically impossible condition.

CONTAM provides two constant flow elements: one for constant mass flow and one for constant volumetric flow.

Variable Flow Fans

The theory of flows induced by fans is summarized in Chapter 18 of the 2000 ASHRAE HVAC Systems and Equipment Handbook [ASHRAE 2004]. More extensive treatment is given in [Osborne 1977]. Fan performance is normally characterized by a performance curve which relates the total pressure *rise* to the flow rate for a given fan speed and air density. Conversion to another fan speed or density is done with the fan laws.

$$Q / Q_0 = N / N_0 \tag{53}$$

or

$$F / F_0 = N\rho / N_0 \rho_0 \tag{54}$$

and

$$P / P_0 = N^2 \rho / N_0^2 \rho_0 \tag{55}$$

where

Q = volume flow rate,
F = mass flow rate,
P = total pressure rise,
ρ = density,
N = rotational speed, and
subscript $_0$ indicates values at the rating conditions for the fan.

These laws are valid if all flow conditions at the two speeds are similar. In particular, they will not apply at very low flows where fully turbulent conditions have not been developed.

In CONTAM the fan performance curve is represented by a cubic polynomial:

$$P = a_0 + a_1 F + a_2 F^2 + a_3 F^3 \tag{56}$$

with

$$P' = \frac{dP}{dF} = a_1 + 2a_2 F + 3a_3 F^2 \tag{57}$$

The polynomial coefficients are developed with air density and fan speed at the rating conditions. Therefore, when N and ρ are not at the rating conditions, convert the actual pressure rise P to P_0.

$$P_0 = P N_0^2 \rho_0 / N^2 \rho \tag{58}$$

Solve (56) for F_0using iterative or analytic solution and (57) for P'_0. Convert these values to current conditions.

$$F = F_0 N\rho / N_0 \rho_0 \tag{59}$$

and

$$P' = P_0 N / N_0 \tag{60}$$

The derivative for flow with respect to pressure drop is given by

$$\frac{dF}{dP} = -\frac{1}{P'} \tag{61}$$

There are two important factors to note on the shape of the fan performance curve. First, it is described by a relationship of the form *P(F)* instead of *F(P)* which would be more appropriate for the calculation of flow and partial derivatives for the Jacobian. The basic shape of the performance curve cannot be well represented by a simple polynomial with *P* as the independent variable. CONTAM uses an analytic solution of the cubic polynomial (56) to determine *F* as a function of *P*.

Second, it is common for the performance curve to contain points of contraflecture (where *P'* = 0) creating up to three different flow rates at certain values of fan pressure rise. This causes difficulty in solving for the flow rate and has points where *dF*/*dP* goes to infinity. However, it is usually not recommended that the fan operate in the region of the contraflecture points. Therefore, the fan can be modeled with a performance curve that does include the contraflecture so long as you make sure that the air distribution system has not operated in that region.

You can identify the points of contraflecture from the coefficients of the polynomial. Solving equation (56) for *F* gives:

$$F = \frac{-2a_2 \pm \sqrt{4a_2^2 - 12a_1a_3}}{6a_3} \tag{62}$$

$4a_2^{\ 2} > 12a_1a_3$	There are two real points of contraflecture
$4a_2^{\ 2} = 12a_1a_3$	There is a point of inflection which still has the derivative problem
$4a_2^{\ 2} < 12a_1a_3$	The fan performance curve is monotonic and well-behaved for computing the derivatives for the Jacobian.

Doorways (Large Openings)

The powerlaw and quadratic models allow flow in only one direction at a time. Flows through large openings (e.g. doorways) tend to be more complex with the possibility of flows in opposite directions in different parts of the opening. The temperature and resulting density differences between two rooms may mean that the stack effect causes a positive pressure difference at the top of the doorway and a negative pressure difference at the bottom (or vice versa) allowing a two-way flow in the opening. A summary of research on heat transfer through doorways is presented in [Barakat 1987]. Most research has attempted to develop dimensionless correlations (using Nusselt, Prandlt, and Grashoff numbers) of the form

$$N_d / \Pr = C \cdot Gr_D^b \tag{63}$$

where b is approximately 0.5 and C lies between 0.22 and 0.33 depending on the temperature difference used for the correlation. It has been shown that such a heat transfer is equivalent to an airflow which can be modeled by powerlaw elements by dividing the total opening into several smaller openings having the same total area but configured to properly account for the magnitude and direction of airflows at different heights in the opening.

An alternative approach is to create a single airflow element which accounts for the flow over the entire opening. A simple theory which estimates the stack induced air flow through a large opening in a vertical partition is given in [Brown and Solvason 1962]. This model of a doorway tends to be faster than the multiple opening approach. However, it also complicates the assembly process for the Jacobian matrix because one or two flows may exist. More importantly, development of the doorway element model requires knowledge of the vertical temperature profile used in the node model (here assumed to be constant) in order to compute the pressure difference as a function of height across the opening. This requirement compromises the independence of the modularity of airflow network program.

Multiple Opening Model

By assuming that the air density in each room is constant, the hydrostatic equation is used to relate pressures at various heights in each room:

$$P_{0j} = P_j + \rho_j g\left(h_j - h_0\right) \text{ and } P_{0i} = P_i + \rho_i g\left(h_i - h_0\right) \tag{64}$$

$$P_j(y) = P_{0j} - \rho_j g y \text{ and } P_i(y) = P_{0i} - \rho_i g y \tag{65}$$

where

P_{0j}, P_{0i} pressure in zones j and i at y = 0, the reference elevation of the opening,
ρ_j, ρ_i air densities of zones j and i,
P_j, P_i reference pressures of zones j and i,
h_j, h_i reference elevations of zones j and i, and
h_0 elevation of the center of the opening.

Following [Brown and Solvason 1962] it is assumed that the velocity of the airflow as a function of height is given by the orifice equation:

$$V(y) = C_d\left(\frac{2\left(P_j(y) - P_i(y)\right)}{\rho}\right)^{1/2} \tag{66}$$

where

C_d = discharge coefficient, and

ρ = density of the air going through the opening.

The opening, H high by W wide, may be divided into multiple openings each representing a thin horizontal strip Δy high and W wide. The mass flow through each strip would be

$$F_y = C_d \sqrt{2\rho\Delta P}\, W\, \Delta y \tag{67}$$

The coefficient for the powerlaw model (20) for each sub-opening is thus given by

$$C_0 = \sqrt{2}\, C_d\, W\, \Delta y \tag{68}$$

Two Opening Model

An approximate model can be constructed by considering an opening between two zones which have no flow paths to any other zones. Then the flow in each direction must be equal and $P_{0j} = P_{0i}$. Integration of equation (66) from y=0 to y=H/2 using (65) to compute ΔP gives the flow through the top half of the opening.

$$w = \int_{y=0}^{y=H/2} \rho V W dy = \frac{C_d}{3} W \sqrt{\rho g \Delta\rho H^3} \tag{69}$$

This is equivalent to a simple powerlaw model with

$$C_0 = C_d \sqrt{2}\left(\frac{W\ H}{2}\right) \tag{70}$$

(a simple orifice with one half the total opening area) placed at an elevation of y = 2H/9. The two opening model is completed with an identical opening at y = -2H/9 elevation allowing an equal flow in the opposite direction. Since the orifice is being modeled with simple orifice openings, any additional pressure drop between the zones due to other forces is included in the two opening model. The model becomes less accurate as the neutral plane shifts from the center of the opening.

Single Opening Model

A third approach is to create a single airflow element which accounts for the flow over the entire opening. Begin by defining the neutral height, Y, where the velocity of the air is zero. From equation (66) this must occur when $P_j(y) = P_i(y)$. From equations (65) this must be

$$Y = \frac{P_{0j} - P_{oi}}{g(\rho_j - \rho_i)}\left(\frac{P_{0i} - P_{oj}}{g(\rho_i - \rho_j)}\right) \tag{71}$$

If $|Y| < H/2$, there is two-way airflow through the opening. If $\rho_j = \rho_i$, the neutral height cannot be computed, but, since there is no possibility of two-way flow, the opening can be considered a simple orifice opening.

Define $\Delta\rho \equiv \rho_j - \rho_i$ and a transformed height coordinate $z \equiv y - Y$. Then the pressure difference across the opening is given by

$$P_j(z) - P_i(z) = -gz\Delta\rho \tag{72}$$

The mass flow through the opening above the neutral height is given by

$$F_a = W \int_{z=0}^{z=\frac{H}{2}-Y} \rho_m V \, dz \quad m = j \text{ or } i \tag{73}$$

and the mass flow below the neutral height by

$$F_b = W \int_{z=\frac{H}{2}-Y}^{z=-} \rho_m V \, dz \quad m = j \text{ or } i \tag{74}$$

Whether the subscript m should be j or i depends on the direction of flow. Integration of equations (73) and (74) gives several different solutions for the airflow depending on the value of *Y* and the sign of Δρ. Defining:

$$\begin{aligned} G &\equiv \frac{2}{3} W C_d \left(2g|\Delta P|\right)^{1/2}, \\ f_a &\equiv \left|\frac{H}{2} - Y\right|^{3/2}, \\ f_b &\equiv \left|\frac{H}{2} + Y\right|^{3/2} \end{aligned} \tag{75}$$

gives the following equations for flows.

Case	Δρ < **0**	Δρ < **0**	
$Y \le -H/2$	$F = -G\sqrt{\rho_i}\,\lvert f_z - f_b \rvert$	$F = G\sqrt{\rho_j}\,\lvert f_z - f_b \rvert$	(76)
$Y \ge H/2$	$F = G\sqrt{\rho_j}\,\lvert f_z - f_b \rvert$	$F = -G\sqrt{\rho_i}\,\lvert f_z - f_b \rvert$	(77)
$-H/2 < Y < H/2$	$F_a = -G\sqrt{\rho_j}\,f_a$	$F_a = G\sqrt{\rho_j}\,f_a$	(78)
	$F_b = G\sqrt{\rho_j}\,f_b$	$F_b = -G\sqrt{\rho_j}\,f_b$	(79)

8 REFERENCES

ACM 1982. ACM-Transactions. Mathematical Software, Vol.8, No. 2, June 1982, p. 190.

Achakji, G.Y. & G.T. Tamura, 1988. "Pressure Drop Characteristics of Typical Stairshafts in High rise Buildings," *ASHRAE Transactions*, 94(1): 1223-1236.

AIVC 1991. *Technical Note AIVC 34 – Airflow Patterns within Buildings: Measurement Techniques*, Coventry UK.

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

ASHRAE 1999a. Method of Testing General Ventilation Air-Cleaning Devices for Removal Efficiency by Particle Size (ANSI approved). *Standard 52.2-1999*. Atlanta, GA.

ASHRAE 1999b. *ASHRAE Handbook - HVAC Applications*, Atlanta GA.

ASHRAE 2001. *ASHRAE Handbook - 2001 Fundamentals*, Atlanta GA.

ASHRAE 2002. *Duct Fitting Database*, v. 2.2.5, Atlanta GA.

ASHRAE 2004. *ASHRAE Handbook - HVAC Systems and Equipment*, Atlanta GA.

ASHRAE 2005. *ASHRAE Handbook - 2005 Fundamentals*, Atlanta GA.

ASTM 1999. Standard Test Method for Determining Air Leakage Rate by Fan Pressurization. *Standard E779-99*. American Society for Testing and Materials, West Conshohocken, PA.

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

Axley, J.W. 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.).

Axley, J.W. 1991. "Adsorption Modeling for Building Contaminant Dispersal Analysis", *Indoor Air*, 1:147-171.

Axley, J.W. 1995. "New Mass Transport Elements and Components for the NIST IAQ Model", NIST GCR 95-676, National Institute of Standards and Technology(U.S.).

Axley, J. W. 2001. Residential Passive Ventilation Systems: Evaluation and Design. AIVC Technical Note 54. International Energy Agency, Coventry.

Baker, P.H., S. Sharples, & I.C. Ward. 1987. "Air Flow through Cracks," *Building and Environment*, Pergamon, 22(4): 293-304.

Barakat, S.A. 1987. "Inter-zone Convective Heat Transfer in Buildings: a Review," *ASME Journal of Solar Engineering*, Vol. 109, May.

Blevins, R.D. 1984. *Applied Fluid Dynamics Handbook*, New York: Van Nostrand Reinhold.

Brown, W.G., & K.R. Solvason. 1962. "Natural Convection through Rectangular Openings in Partitions - 1: Vertical Partitions," *International Journal of Heat and Mass Transfer*, Vol. 5, pp. 859-867.

Clarke, J.A. 1985. *Energy Simulation in Building Design*, Adam Hilger Ltd., Briston and Boston, p 204.

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

Dennis, J.E., Jr. and R.B. Schnabel. 1996. *Numerical Methods for Unconstrained Optimization and Nonlinear Equations*, Society for Industrial and Applied Mathematics, Philadelphia.

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

Dols, W.S., Walton G.N., & Denton, K.R. 2000. "CONTAMW 1.0 User Manual", NISTIR 6476, National Institute of Standards and Technology.

Dols, W.S., L. Wang, S.J. Emmerich, B.J. Polidoro. 2014. "Development and application of an updated whole-building coupled thermal, airflow and contaminant transport simulation program (TRNSYS/CONTAM)." Journal of Building Performance Simulation: 1-21.

Dols, W.S., S.J. Emmerich, B.J. Polidoro. 2014. Evaluating Building Ventilation Strategies Using Newly-Developed TRNSYS/CONTAM Simulation Capabilities. 2014 CIBSE ASHRAE Technical Symposium. Dublin, Ireland, CIBSE.

Emmerich, S.J. and A.K. Persily. 1996. "Multizone Modeling of Three Residential Indoor Air Quality Control Options", NISTIR 5801, National Institute of Standards and Technology.

Emmerich, S.J. and A.K. Persily. 1998. "Energy Impacts of Infiltration and Ventilation in U.S. Office Buildings Using Multizone Airflow Simulation" Proceeding of *IAQ and Energy 98*. New Orleans, Louisiana: ASHRAE.

Emmerich, S.J. and S.J. Nabinger. 2000. "Measurement and Simulation of the IAQ Impact of Particle Air Cleaners in a Single-Zone Building", NISTIR 6461, National Institute of Standards and Technology.

Emmerich, S.J., JE Gorfain, M. Huang, C. Howard-Reed. 2003. "Air and Pollutant Transport from Attached Garages to Residential Living Spaces," NISTIR 7072, National Institute of Standards and Technology, Gaithersburg, MD. December 2003.

EPA 1989. *Exposure Factors Handbook*. Publication number EPA/600/8 89/043.

Fang, J.B. and A.K. Persily. 1995. "Computer Simulations of Airflow and Radon Transport in Four Large Buildings", NISTIR 5611, National Institute of Standards and Technology.

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

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

Guglielmetti, R., D. Macumber, N. Long. 2011. OpenStudio: An Open Source Integrated Analysis Platform. Building Simulation 2011. Sydney, IBPSA.

Hindmarsh, A.C., R. Serban. 2012. *User Documentation for CVODE v2.7.0*. UCRL-SM-20818. Lawrence Livermore National Laboratory.

Howard-Reed, C., Begley, E., Polidoro, B., Dols, W.S. "VOC Source Emission Rate Databases: Model Input and Data Format Issues," *ASTM Conference on Indoor Emissions Testing: Methods and Interpretation*, October 2004.

Howard-Reed, C. and Polidoro, B. "Database Development for Modeling Emissions and Control of Air Pollutants from Consumer Products, Cooking, and Combustion," NISTIR 7364, 2006.

Isaacs, L.T. and 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. 1982. "A Computer Program for Analysis of Smoke Control Systems", NBSIR 82-2512, National Bureau of Standards (U.S.).

Klote, J.H. and J.A. Milke. 2002. *Principles of Smoke Management*, ASHRAE, Atlanta GA.

Klote, J.H., J.A. Milke, P.G. Turnbull, A. Kashef, M.J. Ferreira. 2012. *Handbook of Smoke Control Engineering*. Atlanta, GA, American Society of Heating, Refrigerating and Air-Conditioning Engineers.

Kowalski, W., W. Bahnfleth, D. Witham, B. Severin, and T. Whittam. 2000. Mathematical modeling of ultraviolet germicidal irradiation for air disinfection. *Quantitative Microbiology*. 2(3): p. 249-270.

Lau, J. 2009. Lamp and In-Duct Device Modeling for UVGI System Performance Prediction, Ph.D. Thesis in Architectural Engineering. 2009. The Pennsylvania State University.

Lau, J., W. Bahnfleth, and J. Freihaut. 2009. Estimating the effects of ambient conditions on the performance of UVGI air cleaners. *Building and Environment*. 44(7): p. 1362-1370.

Lee, B. and W.P. Bahnfleth. 2013. Effects of installation location on performance and economics of in-duct ultraviolet germicidal irradiation systems for air disinfection. Building and Environment. 67(0): p. 193-201.

Lorenzetti, D. M., W.S. Dols, A.K. Persily, M.D. Sohn. 2013. "A stiff, variable time step transport solver for CONTAM." Building and Environment 67(0): 260-264.

MODELISAR. 2010. Functional Mock-up Interface for Co-Simulation. ITEA2-07006. MODELISAR consortium. https://www.fmi-standard.org.

MODELISAR. 2010. Functional Mock-up Interface for Model Exchange. ITEA2-07006. MODELISAR consortium. https://www.fmi-standard.org.

Musser, A. and G. Yuill. 1999. "Comparison of Residential Air Infiltration Rates Predicted by Single-Zone and Multizone Models," *ASHRAE Transactions*. Vol. 105(Part 1).

Musser, A. 2000. "Multizone Modeling as an Indoor Air Quality Design Tool", *Proceedings of Healthy Buildings 2000*. Espoo, Finland.

Nouidui, T. S., M. Wetter, W. Zuo. 2014. "Functional mock-up unit for co-simulation import in EnergyPlus." Journal of Building Performance Simulation 7(3): 1-11.

NREL. 2015. "OpenStudio." Retrieved June 5, 2015, from https://www.openstudio.net/. National Renewable Energy Laboratory. Golden, CO.

Osborne, W.C., 1977. *Fans*, 2nd edition, Pergamon Press.

Patankar, S.V,. 1980. *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing. ISBN: 0-89116-522-3.

Persily, A.K. 1998. "A Modeling Study of Ventilation, IAQ and Energy Impacts of Residential Mechanical Ventilation", NISTIR 6162, National Institute of Standards and Technology.

Persily, A.K. and S.R. Martin 2000. "A Modeling Study of Ventilation in Manufactured Houses", NISTIR 6455, National Institute of Standards and Technology.

Persily, A.K. and E.M. Ivy. 2001. "Input Data for Multizone Airflow and IAQ Analysis", NISTIR 6585, National Institute of Standards and Technology.

Press, W.H., S.A. Teukolsky, W.T. Vetterling & B.P. Flannery. 1992. *Numerical Recipes in C: the Art of Scientific Computing, Second Edition*, Cambridge University Press.

Roulet, C-A, and L. Vandaele, *Airflow Patterns Within Buildings Measurement Techniques*, The Air Infiltration and Ventilation Centre, Technical Note 34, 1991, pp. iii.10 and iv.3.

Stoecker, W.F. and P.A. Stoecker. 1989. *Microcomputer Control of Thermal and Mechanical Systems*. Van Nostrand Reinhold, New York.

U.S. DOE. 2013. EnergyPlus External Interface(s) Application Guide (version 8.2). Champaign, IL, University of Illinois.

Versteeg, H.K., and W. Malalasekera, 1995. *An Introduction to Computational Fluid Dynamics : The Finite Volume Method*, Pearson Education Ltd. ISBN: 0-582-21884-5.

Walker, I.S., and D.J. Wilson 1994. "Practical Methods for Improving Estimates of Natural Ventilation Rates", Proceedings 15th AIVC Conference, Vol. 2, pp. 517-525.

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

Walton, G.N., 1989b. "Airflow Network Models for Element-Based Building Airflow Modeling", *ASHRAE Transactions*, Vol. 95, Pt. 2.

Walton, G.N. 1997. "CONTAM96 User Manual", NISTIR 6056, National Institute of Standards and Technology.

Walton, G.N. and W.S. Dols. 2003. "CONTAM 2.1 Supplemental User Guide and Program Documentation," NISTIR 7049, National Institute of Standards and Technology.

Wang, L. (2007). Coupling of Multizone and CFD Programs for Building Airflow and Contaminant Transport Simulations. Lafayette, Purdue University. PhD Thesis.

Wang, L. and Q. Chen (2007). "Theoretical and Numerical Studies of Coupling Multizone and CFD Models for Building Air Distribution Simulations." Indoor Air 17: 348-361.

Wang, L. and Q. Chen (2007). "Validation of a Coupled Multizone and CFD Program for Building Airflow and Contaminant Transport Simulations." HVAC&R Research 13(2): 267-281.

Wang, L. (2008). "Applications of a Coupled Multizone and CFD Model to Calculate Airflow and Contaminant Dispersion in Built Environment for Emergency Management." HVAC&R Research 14(6): 925-939.

Wang, L. (2010). "Using CFD Capabilities of CONTAM 3.0 for Simulating Airflow and Contaminant Transport in and around Buildings." HVAC&R Research 16(6).

Weber, D.D, and R.J. Kearney, 1980. *Natural Convective Heat Transfer Through an Aperture in Passive Solar Heated Buildings*, 5th National Passive Solar Conference, pp. 1037-1041.

Wetter, M. 2011. "Co-Simulation of Building Energy and Control Systems with the Building Controls Virtual Test Bed." Journal of Building Performance Simulation 4 (3):185-203.

Wang, L., W.S. Dols and Q. Chen. 2010. An Introduction to the CFD Capabilities in CONTAM 3.0. SimBuild 2010 - Fourth International Conference of IBPSA-USA. New York City, NY, IBPSA.

Wen, C.Y. and L.T. Fan, 1975. *Models for Flow Systems and Chemical Reactors*, Marcel Dekker, Inc. New York. ISBN: 0-8247-6346-7.

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

APPENDIX A - PRJ FILE FORMAT

CONTAM consists of two separate programs: ContamW provides the graphic user interface for describing the building and viewing simulation results; ContamX performs the simulation. "Project" files (.PRJ extension) provide input to both ContamW and ContamX. Project files are created by the functions in file prjsave.c in ContamW and read by the functions in file prjread.c in ContamW and ContamX. The project file contains some information that is used only by ContamW and not by ContamX, e.g., input units and the SketchPad display data. These data are read when the macro name CTMW is defined to be 1, and they are not read when it is defined to be 0.

Most of the data are read into structures that are defined in the files contam.h and celmts.h for ContamW and files simdat.h and selmts.h for ContamX. Some data are read into global variables that are defined in cglob.h and cxtrn.h for ContamW and sglob.h and sxtrn.h for ContamX. These variables are indicated by a leading underscore, _, in their name.

One important feature of the project files is the tremendous variety of data stored. It has been divided into sections grouping similar kinds of data, or objects, together. Each section is terminated with the special value –999 which serves as a check for some reading errors. Within each section there are usually multiple objects, each stored into an individual structure. Such a section begins with the number of objects and starts the data for each object with a sequence number that serves to check for reading errors. These objects may be ordered as arrays when the total number is constant, or as linked lists to allow the number to vary.

In the following material individual variables are briefly described. Units are indicated within brackets, [ ]. The type of variable is in parentheses (matching the definitions in the include file types.h).

I1 – a one-byte signed integer (C type "char")

I2 – a two-byte signed integer (C type "short")

I4 – a four-byte signed integer (C type "long")

IX – a default length signed integer (C type "int")

UX – a default length unsigned integer (C type "unsigned")

R4 – a four-byte real (C type "float")

Variables used only by ContamW are indicated by {W}. Items that appear in red are the most recently modified items including those whose data size has been increased from 2-bytes to 4-bytes to allow for an increased number of building components. This change took place between versions 2.3 and 2.4.

Comments are included in the project file to make it somewhat human-readable. The exclamation mark, "!" serves as a comment character. Anything after an exclamation mark to the end of the line is not read by the input processor. Any user-defined comments within a PRJ file will not be maintained when the file is saved by ContamW.

Sections of the PRJ File

Section 1: Project, Weather, Simulation, and Output Controls
Section 2: Species and Contaminants
Section 3: Level and Icon Data
Section 4: Day Schedules
Section 5: Week Schedules
Section 6: Wind Pressure Profiles
Section 7: Kinetic Reactions
Section 8a: Filter Elements
Section 8b: Filters
Section 9: Source/Sink Elements
Section 10: Airflow Elements
Section 11: Duct Elements
Section 12a: Control Super Elements
Section 12b: Control Nodes
Section 13: Simple Air Handling System (AHS)
Section 14: Zones
Section 15: Initial Zone Concentrations
Section 16: Airflow Paths
Section 17: Duct Junctions
Section 18: Initial Junction Concentrations
Section 19: Duct Segments
Section 20: Source/Sinks
Section 21: Occupancy Schedules
Section 22: Exposures
Section 23: Annotations

Section 1: Project, Weather, Simulation, and Output Controls

The initial section of the project file is read by the run_read( ) function and saved by the run_save( ) function.

The first line of the project file identifies the source program:

```
_string[] // program name "ContamW" (I1)
_string[] // program version "3.2" (I1)
_echo // > 0 echo input files, > 1 echo bridge messages to the log file (I2)
```

The second line is a user-defined description of the project:

```
_prjdesc[] // (unused) project description (I1) {W}
```

The next line sets the following miscellaneous values:

```
_skheight // total SketchPad height [cells] (I2) {W}
_skwidth // total SketchPad width [cells] (I2) {W}
_def_units // default units: 0 = SI, 1 = US (I2) {W}
_def_flows // default flows: 0 = mass, 1 = volume (I2) {W}
_def_T // default temperature for zones [K] (R4) {W}
_udefT // units for temperature (I2) {W}
_rel_N // angle to true north [degrees] (R4) {W}
_wind_H // elevation for reference wind speed [m] (R4) {W}
```

```
_uwH // units for _wind_H (I2) {W}
_wind_Ao // local terrain constant (R4) {W}
_wind_a // velocity profile exponent (R4) {W}
```

The next line provides SketchPad pseudo-geometry information: {CONTAM 3.1}

```
scale // cell scaling factor (R4)
uScale // units of scaling factor: 0 = m, 1 = ft (I2)
orgRow // row of origin (I2)
orgCol // column of origin (I2)
invYaxis // 0 = positive above origin, 1 = negative above origin (I2)
showGeom // 0/1 = show pseudo-geometry info in status bar (I2)
```

The next line defines the weather (WTHDAT) for steady-state simulation in ContamX:

```
Tambt // ambient temperature [K] (R4)
barpres // barometric pressure [Pa] NOT corrected to sea level (R4)
windspd // wind speed [m/s] (R4)
winddir // wind direction: 0 = N, 90 = E, 180 = S, ...; (R4)
relhum // relative humidity: 0.0 to 1.0 (R4)
daytyp // type of day (1-12) (I2)
uTa // units for Tambt (I2) {W}
ubP // units for barpres (I2) {W}
uws // units for windspd (I2) {W}
uwd // units for winddir (I2) {W}
```

The next line defines weather data (WTHDAT) for the wind pressure test in ContamW:

```
Tambt // ambient temperature [K] (R4) {W}
barpres // barometric pressure [Pa] NOT corrected to sea level (R4) {W}
windspd // wind speed [m/s] (R4) {W}
winddir // wind direction: 0 = N, 90 = E, 180 = S, ...; (R4) {W}
relhum // relative humidity: 0.0 to 1.0 (R4) {W}
daytyp // type of day (1-12) (I2) {W}
uTa // units for Tambt (I2) {W}
ubP // units for barpres (I2) {W}
uws // units for windspd (I2) {W}
uwd // units for winddir (I2) {W}
```

The next four lines define the weather, contaminant and controls files, respectively:

```
_WTHpath[_MAX_PATH] // full name of weather file (I1)
_CTMpath[_MAX_PATH] // full name of contaminant file (I1)
_CVFpath[_MAX_PATH] // full name of continuous values file (I1)
_DVFpath[_MAX_PATH] // full name of discrete values file (I1)
```

The following lines define path location data (PLDDAT) for the creating the WPC file:

three lines for file paths and WPC description:

```
WPCfile[_MAX_PATH] // full name of WPC file (I1)
EWCfile[_MAX_PATH] // full name of EWC data source file (I1) {W}
WPCdesc[] // WPC description (I1) {W}
```

next a line describing reference coordinates:

```
X0 // X-value of ContamW origin in EWC coordinates [m] (R4) {W}
Y0 // Y-value of ContamW origin in EWC coordinates [m] (R4) {W}
```

```
Z0 // Z-value of ContamW origin in EWC coordinates [m] (R4) {W}
angle // Rotation of ContamW relative to EWC coordinates (R4) {W}
u_XYZ // units of coordinates (I2) {W}
```

then some EWC to WPC conversion parameters:

```
epsPath // tolerance for matching path locations [-] (R4) {W}
epsSpcs // tolerance for matching species [-] (R4) {W}
tShift // time shift of EWC data {W} [s] (hh:mm:ss → IX) {W}
dStart // date WPC data starts (I2) {W}
dEnd // date WPC data ends (I2) {W}
_useWPCwp // if true, use WPC file wind pressures (I2)
_useWPCmf // if true, use WPC file mass fractions (I2)
wpctrig // 0/1 = trigger airflow calculation by WPC data (I2) {CONTAM 3.1}
```

The next line defines the location (LOCDAT) (for future use with thermal simulation):

```
latd // latitude (degrees: north +, south -) (R4)
lgtd // longitude (degrees: east +, west -) (R4)
Tznr // time zone (Greenwich = 0, Eastern = -5, etc.) (R4)
altd // elevation above sea level [m] (R4)
Tgrnd // ground temperature [K] (R4)
utg // units for ground temperatures (I2)
u_a // units for elevation (I2)
```

The remaining data is stored in the run control (RCDAT) structure. In ContamX some values may be transferred to other variables before being used. The next two lines control the airflow simulation – first the nonlinear part:

```
sim_af // airflow simulation: 0 = steady, 1 = dynamic (I2)
afcalc // N-R method for non-linear eqns: 0 = SUR, 1 = STR (I2)
afmaxi // maximum number of N-R iterations (I2)
afrcnvg // relative airflow convergence factor (R4)
afacnvg // absolute airflow convergence factor [1/s] (R4)
afrelax // flow under-relaxation coefficient (for SUR) (R4)
uac2 // units for afacnvg (I2)
Pres  // pressure test pressure (R4) {Contam 2.4}
uPres  // units of Pres (I2) {Contam 2.4}
```

and then the linear part:

```
afslae // method for linear equations: 0 = SKY, 1 = PCG (I2)
afrseq // if true, resequence the linear equations (I2)
aflmaxi // maximum number of iterations (PCG) (I2)
aflcnvg // relative convergence factor for (PCG) (R4)
aflinit // if true, do linear airflow initialization (I2)
Tadj // if true, use temperature adjustment (I2)
```

The next four lines control the linear solver for contaminant calculations:

first for cyclic simulation:

```
sim_mf // mass fraction (contaminant) simulation:
 // 0 = none, 1 = steady, 2 = transient, 3 = cyclic (I2)
ccmaxi // simulation: maximum number of cyclic iterations (I2)
ccrcnvg // relative convergence factor (R4)
```

```
ccacnvg // absolute convergence factor [kg/kg] (R4)
ccrelax // (unused) over-relaxation coefficient (R4)
uccc // units for ccacnvg (I2)
```

then for non-trace contaminants:

```
mfnmthd // simulation: 0 = SKY, 1 = BCG, 2 = SOR, 3 = LU (I2)
mfnrseq // if true, resequence the linear equations (I2)
mfnmaxi // maximum iterations (I2)
mfnrcnvg // desired relative convergence (R4)
mfnacnvg // desired absolute convergence (R4)
mfnrelax // relaxation coefficient (R4)
mfngamma // trapezoidal integration factor (R4)
uccn // units for mfnacnvg (I2)
```

then for trace contaminants:

```
mftmthd // 0 = SKY, 1 = BCG, 2 = SOR, 3 = LU (I2)
mftrseq // if true, resequence the linear equations (I2)
mftmaxi // maximum iterations (I2)
mftrcnvg // desired relative convergence (R4)
mftacnvg // desired absolute convergence (R4)
mftrelax // relaxation coefficient (R4)
mftgamma // trapezoidal integration factor (R4)
ucct // units for mftacnvg (I2)
```

then for CVODE contaminant solver: {CONTAM 3.1}

```
mfvmthd // 0 = SKY, 2 = SOR, 3 = LU (1 n/a) (I2)
mfvrseq // if true, resequence the linear equations (I2)
mfvmaxi // maximum iterations (I2)
mfvrcnvg // desired relative convergence (R4)
mfvacnvg // desired absolute convergence (R4)
mfvrelax // relaxation coefficient (R4)
uccv // units for mfvacnvg (I2)
```

The next line sets the integration method and short time step contaminant calculation parameters:

```
mf_solver // mass fraction integration method: {3.1}
          // 0=trapezoid, 1=STS, 2=CVODE (I2)
sim_1dz // if true, use 1D zones (I2)
sim_1dd // if true, use 1D ducts (I2)
celldx // default length of duct cells for C-D model [m] (R4)
sim_vjt // if true, compute variable junction temperatures (I2)
udx // units of celldx (I2)
```

The next line contains CVODE contaminant solver parameters: {CONTAM 3.1}

```
cvode_mth // 0 = am, 1 = bdf (I2)
cvode_rcnvg // relative convergence factor(R4)
cvode_acnvg // absolute convergence factor(R4)
cvode_dtmax // maximum time step (R4)
```

The next line contains density control parameters:

```
tsdens // (0/1) vary density during time step (I2)
```

```
tsrelax // (inactive) under-relaxation factor for calculating dM/dt (R4)
tsmaxi // maximum number of iterations for density changes (I2)
cnvgSS // (0/1) converge density in steady state init (I1) {2.4b}
densZP // (0/1) density = f(zone pressure) (I1) {2.4b, 3.1}
stackD // (0/1) density varies hydrostatically (I1) {2.4b, 3.1}
dodMdt // (0/1) include dM/dt in airflow calc (I1) {2.4b, 3.1}
```

The next line sets the dates, times, and time steps for simulation:

```
date_st // day-of-year to start steady simulation (mmmdd → IX)
time_st // time-of-day to start steady simulation (hh:mm:ss → I4)
date_0 // day-of-year to start transient simulation (mmmdd → IX)
time_0 // time-of-day to start transient simulation (hh:mm:ss → I4)
date_1 // day-of-year to end transient simulation (mmmdd → IX)
time_1 // time-of-day to end transient simulation (hh:mm:ss → I4)
time_step // simulation time step [s] (hh:mm:ss → IX)
time_list // simulation output (results) time step [s] (hh:mm:ss → IX)
time_scrn // simulation status time step [s] (up to 1 day) (hh:mm:ss → I4)
```

The next line controls the use of the restart file:

```
restart // use restart file (I2)
rstdate // restart date (mmmdd → IX)
rsttime // restart time (hh:mm:ss → I4)
```

The next line controls output of simulation result files.

```
_list // data dump parameter (I2)
      // > 0 dump matrix analysis,
      // = 2 dump SIM file output,
      // > 2 dump lognotes.
_doDlg // (0/1) ContamX will display dialog box (I1)
_pfsave // (0/1) save path flow results to SIM file (I1)
_zfsave // (0/1) save zone flow results to SIM file (I1)
_zcsave // (0/1) save mass fraction results to SIM file (I1)
_achvol // (0/1) ACH based on true volumes instead of std volumes
_achsave // (0/1) save building air exchange rate transient data
_abwsave // (0/1) save air exchange rate box-whisker data
_cbwsave // (0/1) save contaminant box-whisker data
_expsave // (0/1) save exposure transient data
_ebwsave // (0/1) save exposure box-whisker data
_zaasave // (0/1) save zones age-of-air transient data
_zbwsave // (0/1) save zones age-of-air box-whisker data
```

The next line controls output of more simulation result files: {CONTAM 3.1}

```
rzfsave // (0/1) save binary zone flow file
rzmsave // (0/1) save binary avg zone mass fraction file
rz1save // (0/1) save binary 1D zone cell mass fraction file
csmsave // (0/1) save text contaminant summary file
srfsave // (0/1) save text surface result file
logsave // (0/1) save text controls log file
```

The next line controls output of more simulation result files: {CONTAM 3.2}

```
bcexsave // (0/1) save basic cex file
dcexsave // (0/1) save detailed cex files
pfsqlsave // (0/1) save path flow results to SQL
zfsqlsave // (0/1) save zone flow results to SQL
zcsqlsave // (0/1) save zone mass (contaminant) results to SQL
```

The next line provides a set of integers that are used for development purposes: {CONTAM 3.1}

```
save[0-15] // (unused by CONTAM; subject to change without notice) (I1)
```

The next two lines contain a list of real values provided for PRJ file flexibility. They are stored in the rvals[] vector {CONTAM 2.4b}

```
nrvals  // number of real values in the vector (I2)
rvals[0]  // standard density for air change calculation  [kg/m3](R4)
rvals[1]  // acceleration due to gravity  [m/s2](R4)
```

The next line controls which sections are output to the .val file: {CONTAM 2.4a}

```
BldgFlowZ // output building airflow test (zones) (IX)
BldgFlowD // output building airflow test (ducts) (IX)
BldgFlowC // output building airflow test (classified flows) (IX)
```

The next line controls cfd calculations and coupling: {CONTAM 3.0}

```
cfd_ctype // (0=no cfd, 1=post, 2=quasi, 3=dynamic) cfd coupling method (I2)
cfd_convcpl // convergence factor for dynamic coupling (R4)
cfd_var // (0/1) use .var file (I2)
cfd_zref // currently not used (I2)
cfd_imax // max number of dynamic coupling iterations (I2)
cfd_dtcmo // number of iterations between outputs to .cmo file (I2)
```

The run control section is terminated with:

```
-999 // used to check for a read error in the above data
```

Note on outputs: _pfsave and _zfsave always have the same value – 0 or 1.

Section 2: Species and Contaminants

Species/contaminant data are read by the spcs_read( ) function and saved by the spcs_save( ) function in ContamW. They are read by ctm_read( ) in ContamX.

The species/contaminant section starts with:

```
_nctm // total number of species simulated (= contaminants) (I2)
```

The next line lists the numbers of the _nctm species treated as contaminants. The ordering of the contaminants is important, and is set in ContamW.

```
_ctm[i] // i = 0 to nctm-1
```

The third line of this section gives:

```
_nspcs // the number of species (IX)
```

This is followed by a header line and then two lines of data for each species:

The first line of species data consists of:

```
  nr // species number (IX), in order from 1 to _nspcs
  sflag // 1 = simulated, 0 = unsimulated species (I2) {W}
  ntflag // 1 = non-trace, 0 = trace species (I2) {W}
  molwt // molar mass [kg/kmol] - gas (R4)
```

```
mdiam // mean diameter – particle [m] (R4)
edens // effective densiity – particle [kg/m^3] (R4)
decay // decay constant [1/s] (R4) {W}
Dm // molecular diffusion coefficient [m²/s] (R4)
ccdef // default concentration [kg/kg air] (R4)
Cp // (unused) specific heat at constant pressure [J/kgK] (R4)
Kuv // UVGI susceptability constant [m2/J] (R4) {CONTAM 3.2}
ucc // units to display concentration (I2) {W}
umd // units to display mean diameter (I2) {W}
ued // units to display effective density (I2) {W}
udm // units to display diffusion coefficient (I2) {W}
ucp // units to display specific heat (I2) {W}
name[] // species name (I1)
```

The second line is:

```
desc[] // species description (I1) {W}
```

The section is terminated with:

```
-999 // used to check for a read error in the above data (I2)
```

Section 3: Level and Icon Data

The data for recreating the SketchPad is stored in this section. The data are read by the level_read( ) function and saved by the level_save( ) function. The level data are stored in LEV_DATA structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The icon data are stored in ICON_DAT structures in ContamW and are not saved in ContamX.

The level/icons section starts with:

```
nlev // number of levels (IX)
```

This is followed by a data header comment line and then data for all nlev levels.

Each level has a data line that includes:

```
nr // level number (IX), in order from 1 to nlev
refht // reference elevation of level [m] (R4)
delht // delta elevation to next level [m] (R4) {W}
nicon // number of icons on this level (IX)
u_rfht // units of reference elevation (I2) {W}
u_dlht // units of delta elevation (I2) {W}
name[] // level name (I1)
```

This line is followed by a comment line and then data for nicon icons.

Each icon has a data line consisting of:

```
icon // icon type – see 'special symbols' in contam.h (I2) {W}
row // row position on the SketchPad (I2) {W}
col // column position on the SketchPad (I2) {W}
nr // zone, path, duct, etc., number (I2) {W}
```

The section is terminated with:

```
-999 // used to check for a read error in the above data (I2)
```

Section 4: Day Schedules

Day schedule data are read by the dschd_read( ) function and saved by the dschd_save( ) function. The data are stored in the DY_SCHD structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The day schedules section starts with:

```
_ndsch // number of day schedules (IX)
```

This is followed by a data header comment line and then data for all `_ndsch` schedules.

For each schedule the first data line includes:

```
  nr // schedule number (IX); in order from 1 to _ndsch
  npts // number of data points (I2)
  shape // 0 = rectangular; 1 = trapezoidal (I2)
  utyp // type of units (I2) {W}
  ucnv // units conversion (I2) {W}
  name[] // schedule name (I1) {W}
```

and the second line has:

```
  desc[] // schedule description (I1) {W} may be blank
```

This is followed by a line for each of the npts data points:

```
  time // time-of-day [s] (hh:mm:ss converted to I4)
  ctrl // corresponding control value (R4) [-]
```

The section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 5: Week Schedules

Week schedule data are read by the wschd_read( ) function and saved by the wschd_save( ) function. The data are stored in the WK_SCHD structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The week schedules section starts with:

```
_nwsch // number of week schedules (IX)
```

This is followed by a data header comment line and then data for all _nwsch schedules.

For each week schedule the first data line includes:

```
  nr // schedule number (IX); in order from 1 to _nwsch
  utyp // type of units (I2) {W}
  ucnv // units conversion (I2) {W}
  name[] // schedule name (I1) {W}
```

and the second line has:

```
  desc[] // schedule description (I1) {W} may be blank
```

and the third line has:

```
  j1 … j12 // 12 day schedule indices (IX) - converted to pointers
```

The section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 6: Wind Pressure Profiles

Wind pressure profile data are read by the wind_read( ) function and saved by the wind_save( ) function. The data are stored in the WIND_PF structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The wind pressure profiles section starts with:

```
_nwpf // number of wind pressure profiles (IX)
```

This is followed by a data header comment line and then data for all _nwpf profiles.

For each wind pressure profile the first data line includes:

```
  nr // profile number (IX); in order from 1 to _nwpf
  npts // number of data points (I2)
  type // 1 = linear; 2 = cubic spline; 3 = trigonometric (I2)
  name[] // schedule name (I1) {W}
```

and the second line has:

```
  desc[] // profile description (I1) {W} may be blank
```

This is followed by a line for each of the npts data points that are used to compute the cubic spline or trigonometric coefficients for wind pressure profiles:

```
  azm[] // wind azimuth value {R4} [degrees]
  coef[] // normalized wind pressure coefficients {R4} [-]
```

The section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 7: Kinetic Reactions

Kinetic reaction data are read by the kinetic_read( ) function and saved by the kinetic_save( ) function. The data are stored in the KNR_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The kinetic reactions section starts with:

```
_nkinr // number of kinetic reactions (IX)
```

This is followed by a data header comment line and then data for all _nkinr reactions.

For each reaction the first data line includes:

```
  nr // reaction number (IX); in order from 1 to _nkinr
  nkrd // number of reactions (I2)
  name[] // reaction name (I1) {W}
```

and the second line has:

```
  desc[] // reaction description (I1) {W} may be blank
```

This is followed by nkrd lines of reaction data:

```
  prod[] // product species name (I1)
  src[] // source species name (I1)
  coef // reaction coefficient (R4)
```

The kinetic reactions section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 8a: Filter Elements

Filter element data are read by the flte_read( ) function and saved by the flte_save( ) function. The data are stored in the FLT_ELT structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The filter element section starts with:

```
_nflte // number of filter elements (IX)
```

This is followed by data for all `_nflte` filters.

For each element the first data line includes:

```
nr // element number (IX); in order from 1 to _nflte
ftype // element data type (string ? I2)
 // type names are stored in _flte_names in ftype order.
area // face area of filter [m] (R4)
depth // depth of filter [m] (R4)
dens // density of filter material [kg/m^3] (R4)
 // note: area * depth * dens = filter mass [kg]
ual // units of area and depth (I2) {W}
ud // units of density (I2) {W}
name[] // element name (I1) {W}
```

and the second line has:

```
desc[] // filter description (I1) {W} may be blank
```

This is followed by lines of data that depend on the element data type.

The introductory lines that follow give the type number, the program macro defined as that number, the type name from _flte_names, and the data structures to hold the data. The data structures are defined in celmts.h in ContamW and selmts.h in ContamX. Type numbers can change as long as the ordering in _flte_names and related arrays reflect that order.

Element type 0 [FL_CEF] "cef" stored in structure CEF_DAT.

The constant efficiency filter data consist of:

```
nspcs // number of species (IX)
```

This is followed by `nspcs` lines of filter efficiency data:

```
spcs[] // species name (I1)
eff // filter efficiency [-] (R4)
```

Element type 1 [FL_PF0] "pf0" stored in structure PF0_DAT.

The simple particle filter data consist of:

```
npts // number of data points (I2)
usz // units of particle size (I2) {W}
```

This is followed by `npts` lines of filter efficiency data:

```
size // particle size [m] (R4)
eff // filter efficiency [-](R4)
```

Element type 2 [FL_GF0] "gf0" stored in structure GF0_DAT.

The simple gas filter data consiss of:

```
nspcs // number of species (IX)
```

This is followed by `nspcs` sections consisting of:

```
spcs[] // species name (I1)
bthru[] // break through eff (R4) {Contam 2.4}
npts // number of data points (I2)
```

Each of which is followed by `npts` lines of filter efficiency data:

```
load // relative total loading [-] (R4)
eff // filter efficiency for this species[-] (R4)
```

Element type 3 [FL_SPF] "spf" stored in structure SPF_DAT.

The super filter data consist of:

```
nselmt // number of sub-elements(I2)
```

Then a line of data containing a list of the number of each sub-element in flow order:

```
elmt[] // sub-element numbers (IX) - converted to pointers
```

The filter element section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 8b: Filters

Filter data are read by the filter_read( ) function and saved by the filter_save( ) function. The data are stored in the FLT_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The filter section starts with:

```
_nfilt // number of filters (IX)
```

This is followed by data for all `_nfilt` filters.

For each filter the first data line includes:

```
nr // filter number (IX); in order from 1 to _nfilt
fe  // filter element number (IX)
nsub  // number of sub-elements, 1 if not super filter (IX)
```

This is followed by `nsub` lines of initial total relative loadings:

```
load // initial relative loading [-] (R4)
nsload // number of species loadings (I2) [0]
```

This is followed by `nload` lines of species loading data: [not yet used; nsload = 0]

```
spcs[]   // species name (I1)
load     // initial relative loading [-] (R4)
```

Section 9: Source/Sink Elements

Source/sink elements are read by the cselmt_read( ) function and saved by the cselmt_save( ) function. The data are stored in the CSE_DAT structures that are defined in contam.h for ContamW and simdat.h for ContamX. CSE_DAT includes a pointer to the element specific data structure.

The source/sink elements section starts with:

```
_ncse // number of source/sink elements (IX)
```

This is followed by a data header comment line and then data for all _ncse elements.

For each element the first data line includes:

```
nr // element number (IX); in order from 1 to _ncse
spcs[] // species name (I1)
ctype // element data type (string ? I2)
element type names are stored in _cse_dnames in ctype order.
name[] // element name (I1) {W}
```

and the second line has:

```
desc[] // element description (I1) {W} may be blank
```

This is followed by one or more lines of data that depend on the element data type.

The introductory lines that follow give the type number, the program macro defined as that number, the type name from_cse_dnames, and the data structures to hold the data. The data structures are defined in celmts.h in ContamW and selmts.h in ContamX. Type numbers can change as long as the ordering in _csse_names and related arrays reflect that order.

Element type 0 [CS_CCF] "ccf" stored in structure CSE_CCF.

The constant coefficient source model data:

```
G // generation rate [kg/s] (R4)
D // deposition rate [kg/s] (R4)
u_G // units of generation (I2) {W}
u_D // units of deposition (I2) {W}
```

Element type 1 [CS_PRS] "prs" stored in structure CSE_PRS.

The pressure driven source data:

```
G // generation rate [kg/s] (R4)
x // pressure exponent [-] (R4)
u_G // units of generation (I2) {W}
```

Element type 2 [CS_CUT] "cut" stored in structure CSE_CUT.

The concentration cutoff model data:

```
G // generation rate [kg/s] (R4)
Co // cutoff concentration [kg/kg] (R4)
u_G // units of generation (I2) {W}
u_C // units of concentration (I2) {W}
```

Element type 3 [CS_EDS] "eds" stored in structure CSE_EDS.

The exponential decay model data:

```
G0 // initial generation rate [kg/s] (R4)
k // decay constant [1/s] (R4)
u_G // units of generation (I2) {W}
u_k // units of time (I2) {W}
```

Element type 4 [CS_BLS] "bls" stored in structure CSE_BLS.

The boundary layer diffusion model data:

```
hm // average film mass transfer coefficient [m/s] (R4)
rho // film density of air [kg/m^3] (R4)
Kp // partition coefficient (R4)
M // adsorbent mass/unit area [kg/m^2] (R4)
u_h // units of hm (I2) {W}
u_r // units of rho (I2) {W}
u_M // units of M (I2) {W}
```

Element type 5 [CS_BRS] "brs" stored in structure CSE_BRS.

The burst source data:

```
M // mass added to zone in one time step [kg] (R4)
u_M // units of mass (I2) {W}
```

Element type 6 [CS_DVS] "dvs" stored in structure CSE_DVS.

The deposition velocity sink model data:

```
dV        // deposition velocity [m/s] (R4)
dA        // deposition area [m^2] (R4)
u_V       // units of velocity (I2) {W}
u_A       // units of area (I2) {W}
```

Element type 7 [CS_DRS] "drs" stored in structure CSE_DRS.

The deposition rate sink model data:

```
kd        // deposition rate [1/s] (R4)
u_k       // units of deposition rate (I2) {W}
```

Element type 8 [CS_DVR] "dvr" stored in structure CSE_DVR. {CONTAM 3.0}

The deposition velocity resuspension source/sink model data:

```
dV        // deposition velocity [m/s] (R4)
R         // resuspension rate [1/s] (R4)
dA        // deposition surface area [m2] (R4)
rA        // resuspension surface area [m2] (R4)
u_V       // units of deposition velocity (I2) {W}
u_R       // units of resuspension rate (I2) {W}
u_dA      // units of deposition areaa (I2) {W}
u_dR      // units of resuspension area (I2) {W}
```

Element type 11 [CS_PLM] "plm" stored in structure CSE_PLM. {CONTAM 2.4b}

The NRCC power law model data:

```
a  // coefficient(R4) [kg/s]
b  // exponent (R4)
tp  // PL model time [s] (R4)
u_a // units of initial emission factor (I2) {W}
u_tp // units of PL model time (I2) {W}
```

Element type 12 [CS_PKM] "pkm" stored in structure CSE_PKM. {CONTAM 2.4b}

The NRCC peak model data:

```
a  // peak emission rate(R4) [kg/s]
b  // fitting parameter (R4)
tp  // time to peak [s] (R4)
u_a // units of peak emission rate (I2) {W}
u_tp // units of peak time (I2) {W}
```

Element type 13 [CS_SUP] "sup". {CONTAM 2.4b}

The super source element data:

```
n  // number of sub-elements (IX)
```

A line containing a list of all sub-element numbers:

```
elmt[] // sub-element numbers (IX) - converted to pointers
```

The source/sink element section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 10: Airflow Elements

Airflow elements are read by the afelmt_read( ) function and saved by the afelmt_save( ) function. The data are stored in the AFE_DAT structures that are defined in contam.h for ContamW and simdat.h for ContamX. AFE_DAT includes a pointer to the element specific data structure.

The airflow elements section starts with:

```
_nafe // number of airflow elements (IX)
```

This is followed by a data header comment line and then data for all _nafe elements.

For each element the first data line includes:

```
nr // element number (IX); in order from 1 to _nafe
icon // icon used to represent flow path (R4) {W}
dtype // element data type (string ? I2)
      // element type names are stored in _afe_dnames in dtype order.
name[] // element name (I1) {W}
```

and the second line has:

```
desc[] // element description (I1) {W} may be blank
```

This is followed by one or more lines of data that depend on the element data type.

The introductory lines that follow give the type number, the program macro defined as that number, the type name from _afe_dnames, and the data structures to hold the data. The data structures are defined in celmts.h in ContamW and selmts.h in ContamX. Type numbers can change as long as the ordering in _afe_dnames and related arrays reflect that order.

Element type 0 [PL_ORFC] "plr_orfc" stored in structure PLR_ORF.

The orifice data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
area // actual area [m^2] (R4) {X}
dia // hydraulic diameter [m] (R4) {X}
coef // flow coefficient (R4) {X}
Re // laminar/turbulet transition Reynolds number [-](R4) {X}
u_A // units of area (I2) {X}
u_D // units of diameter (I2) {X}
```

Element type 1 [PL_LEAK1] "plr_leak1" stored in structure PLR_LEAK.

Element type 2 [PL_LEAK2] "plr_leak2" stored in structure PLR_LEAK.

Element type 3 [PL_LEAK3] "plr_leak3" stored in structure PLR_LEAK.

The leakage area data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
coef // flow coefficient (R4) {W}
pres // reference pressure drop [Pa] (R4) {W}
area1 // leakage area per item [m^2] (R4) {W}
area2 // leakage area per unit length [m^2/m] (R4) {W}
area3 // leakage area per unit area [m^2/m^2] (R4) {W}
u_A1 // units of area1 [m^2] (I2) {W}
u_A2 // units of area2 [m^2/m] (I2) {W}
u_A3 // units of area3 [m^2/m^2] (I2) {W}
u_dP // units of pressure (I2) {W}
```

Element type 4 [PL_CONN] "plr_conn" stored in structure PLR_CONN.

The (ASCOS compatible) connection data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent - 0.5 (R4)
area // actual area [m^2] (R4) {W}
coef // flow coefficient (R4) {W}
u_A // units of area (I2) {W}
```

Element type 5 [PL_QCN] "plr_qcn" stored in structure PLR_QCN.

The volume flow powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
```

Element type 6 [PL_FCN] "plr_fcn" stored in structure PLR_FCN.

The mass flow powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
```

Element type 7 [PL_TEST1] "plr_test1" stored in structure PLR_TEST1.

The single test point powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
dP // pressure drop [Pa] (R4) {W}
Flow // flow rate [kg/s] (R4) {W}
u_P // units of pressure drop {W}
u_F // units of flow (I2) {W}
```

Element type 8 [PL_TEST2] "plr_test2" stored in structure PLR_TEST2.

The two test points powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
dP1 // point 1 pressure drop [Pa] (R4) {W}
F1 // point 1 flow rate [kg/s] (R4) {W}
dP2 // point 2 pressure drop [Pa] (R4) {W}
F2 // point 2 flow rate [kg/s] (R4) {W}
u_P1 // units of pressure drop (I2) {W}
u_F1 // units of flow (I2) {W}
u_P2 // units of pressure drop (I2) {W}
u_F2 // units of flow (I2) {W}
```

Element type 9 [PL_CRACK] "plr_crack" stored in structure PLR_CRACK.

The crack powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
length // crack length [m] (R4) {W}
width // crack width [m] (R4) {W}
u_L // units of length (I2) {W}
u_W // units of width (I2) {W}
```

Element type 10 [PL_STAIR] "plr_stair" stored in structure PLR_STAIR.

The stairwell powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
Ht // distance between levels [m] (R4) {W}
Area // cross-sectional area [m^2] (R4) {W}
peo // density of people [pers/m^2] (R4) {W}
tread // 1 = open tread 0 = closed {W}
u_A // units of area (I2) {W}
u_D // units of distance (I2) {W}
```

Element type 11 [PL_SHAFT] "plr_shaft" stored in structure PLR_SHAFT.

The shaft powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
Ht // distance between levels [m] (R4) {W}
area // cross-sectional area [m^2] (R4) {W}
perim // perimeter [m] (R4) {W}
rough // roughness [m] (R4) {W}
u_A // units of area (I2) {W}
u_D // units of distance (I2) {W}
u_P // units of perimeter (I2) {W}
u_R // units of roughness (I2) {W}
```

Element type 12 [PL_BDQ] "plr_bdq" stored in structure PLR_BDQ.

The volume flow powerlaw backdraft damper data consist of:

```
lam // laminar flow coefficient {R4}
Cp // turbulent flow coefficient ( dP > 0 ) {R4}
xp // pressure exponent ( dP > 0 ) {R4}
Cn // turbulent flow coefficient ( dP < 0 ) {R4}
xn // pressure exponent ( dP < 0 ) {R4}
```

Element type 13 [PL_BDF] "plr_bdf" stored in structure PLR_BDF.

The mass flow powerlaw backdraft damper data consist of:

```
lam // laminar flow coefficient {R4}
Cp // turbulent flow coefficient ( dP > 0 ) {R4}
xp // pressure exponent ( dP > 0 ) {R4}
Cn // turbulent flow coefficient ( dP < 0 ) {R4}
xn // pressure exponent ( dP < 0 ) {R4}
```

Element type 14 [QFR_QAB] "qfr_qab" stored in structure QFR_QAB.

The volume flow quadratic data consist of:

```
a // dP = a*Q + b*Q*Q {R4}
b // {R4}
```

Element type 15 [QFR_QAF] "qfr_fab" stored in structure QFR_FAB.

The mass flow quadratic data consist of:

```
a // dP = a*F + b*F*F {R4}
b // {R4}
```

Element type 16 [QFR_CRACK] "qfr_crack" stored in structure QFR_CRACK.

The crack mass flow quadratic data consist of:

```
a // dP = a*F + b*F*F {R4}
b // {R4}
length // crack length [m] {R4}
width // crack width [m] {R4}
depth // crack depth [m] {R4}
nB // number of bends
u_L // units of length
u_W // units of width
```

```
u_D // units of depth
```

Element type 17 [QFR_TEST2] "qfr_test2" stored in structure QFR_TEST2.

The two test points mass flow quadratic data consist of:

```
a // dP = a*F + b*F*F {R4}
b // {R4}
dP1 // point 1 pressure drop [Pa] (R4) {W}
F1 // point 1 flow rate [kg/s] (R4) {W}
dP2 // point 2 pressure drop [Pa] (R4) {W}
F2 // point 2 flow rate [kg/s] (R4) {W}
u_P1 // units of pressure drop (I2) {W}
u_F1 // units of flow (I2) {W}
u_P2 // units of pressure drop (I2) {W}
u_F2 // units of flow (I2) {W}
```

Element type 18 [DR_DOOR] "dor_door" stored in structure AFE_DOR.

The single opening doorway data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
dTmin // minimum temperature difference for two-way flow [C] (R4)
      // Not used since version 2.4.
ht // height of doorway [m] (R4)
wd // width of doorway [m] (R4)
cd // discharge coefficient
u_T // units of temperature (I2) {W}
u_H // units of height (I2) {W}
u_W // units of width (I2) {W}
```

Element type 19 [DR_PL2] "dor_pl2" stored in structure DR_PL2.

The double opening doorway data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
dH // distance above | below midpoint [m] {R4}
ht // height of doorway [m] (R4) {W in v. 2.0}
wd // width of doorway [m] (R4) {W}
cd // discharge coefficient [-] (R4) {W}
u_H // units of height (I2) {W}
u_W // units of width (I2) {W}
```

Element type 20 [FN_CMF] "fan_cmf" stored in structure AFE_CMF.

The constant mass flow fan data consist of:

```
Flow // design flow rate [kg/s] (R4)
u_F // units of flow (I2) {W}
```

Element type 21 [FN_CVF] "fan_cvf" stored in structure AFE_CVF.

The constant volume flow fan data consist of:

```
Flow // design flow rate [m^3/s] (R4)
u_F // units of flow (I2) {W}
```

Element type 22 [FN_FAN] "fan_fan" stored in structure AFE_FAN.

The first line of performance curve fan data consists of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
rdens // reference fluid density [kg/m^3] {R4}
fdf // free delivery flow (prise = 0) [kg/s] {R4}
sop // shut-off pressure (flow = 0) [Pa] {R4}
off // fan is off if (RPM/rated RPM) < off {R4}
```

The second line consists of:

```
fpc[4] // fan performance polynomial coefficients {R4}
npts // number of mesaured data points (I2) {W}
Sarea // shut-off orifice area [m^2] {R4} {W}
u_Sa // units of shut-off area (I2) {W}
```

The next npts lines consists of:

```
mF // measured flow rates [kg/s] (R4) {W}
u_mF // units of measured flows (I2) {W}
dP // measured pressure rises [Pa] (R4) {W}
u_dP // units of pressure rises (I2) {W}
rP // revised pressure rises [Pa] (R4) {W}
u_rP // units of revised pressures (I2) {W}
```

Element type 23 [CS_FSP] "csf_fsp" stored in structure AFE_CSF.

Element type 24 [CS_QSP] " csf_qsp " stored in structure AFE_ CSF.

Element type 25 [CS_PSF] " csf_psf " stored in structure AFE_ CSF.

Element type 26 [CS_PSQ] " csf_psq " stored in structure AFE_ CSF.

The first line of cubic spline fit element data consists of:

```
npts // number of data points (I2)
u_x // units for x (I2)
u_y // units for y (I2)
```

followed by npts lines of x, y pairs of data

```
x  // value of independent variable (R4)
y  // value of dependent variable (R4)
```

Element type 27 [AF_SUP] "sup_afe" stored in structure AFE_SUP.

The first line of super airflow element data consists of:

```
nse // number of sub-elements(I2)
sched // scheduled sub-element number (IX) (only one)
u_H // units of relative height (I2)
```

followed by nse lines of sub-element data

```
nr  // sub-element number (IX)
relHt // relative height of sub-element (R4)
filt // 1= filtered, 0= not (I2)
```

The airflow element section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 11: Duct Elements

Duct elements are read by the afelmt_read( ) function with the duct flag set to 1 and saved by the afelmt_save( ) function. The data are stored in the AFE_DAT structures that are defined in contam.h for ContamW and simdat.h for ContamX. AFE_DAT includes a pointer to the element specific data structure.

NOTE: Some of the elements in this section were incorrectly documented in the CONTAM 2.1 program documentation. However, the data structures have not changed for this version, only the documentation provided herein.

The duct elements section starts with:

```
_ndfe // number of duct flow elements (IX)
```

This is followed by a data header comment line and then data for all _ndfe elements.

For each element the first data line includes:

```
  nr // element number (IX); in order from 1 to _nafe
  icon // icon used to represent flow path ((R4) {W}
  dtype // element data type (string ? I2)
        // element type names are stored in _afe_dnames in dtype order.
  name[] // element name (I1) {W}
```

and the second line has:

```
  desc[] // element description (I1) {W} may be blank
```

This is followed by two or more lines of data that depend on the element data type.

The introductory lines that follow give the type number, the program macro defined as that number, the type name from _afe_dnames, and the data structures to hold the data. The data structures are defined in celmts.h in ContamW and selmts.h in ContamX. Type numbers can change as long as the ordering in _afe_dnames and related arrays reflect that order.

Element type 23 [DD_DWC] "dct_dwc" stored in structure DEF_DWC.

The Darcy-Colebrook data consist of:

```
  rough // roughness dimension [m] (R4)
  lam // laminar total loss coefficient per [m] of duct (R4)
  u_R // units for roughness (I2) {W}
```

Element type 24 [DD_PLR] "dct_plr" stored in structure DFE_ORF.

The orifice data consist of:

```
  lam // laminar flow coefficient (R4)
  turb // turbulent flow coefficient (R4)
  expt // pressure exponent (R4)
  area // actual area [m^2] (R4) {X}
  dia // hydraulic diameter [m] (R4) {X}
  coef // flow coefficient (R4) {X}
  u_A // units of area (I2) {X}
```

```
u_D // units of diameter (I2) {X}
```

Element type 26 [DD_QCN] "dct_qcn" stored in structure DFE_QCN.

The volume flow powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
```

Element type 25 [DD_FCN] "dct_fcn" stored in structure DFE_FCN.

The mass flow powerlaw data consist of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
```

Element type 28 [DD_CMF] "dct_cmf" stored in structure DFE_CMF.

The constant mass flow fan data consist of:

```
Flow // design flow rate [kg/s] (R4)
u_F // units of flow (I2) {W}
```

Element type 29 [DD_CVF] "dct_cvf" stored in structure DFE_CVF.

The constant volume flow fan data consist of:

```
Flow // design flow rate [m^3/s] (R4)
u_F // units of flow (I2) {W}
```

Element type 27 [DD_FAN] "dct_fan" stored in structure DFE_FAN.

The first line of performance curve fan data consists of:

```
lam // laminar flow coefficient (R4)
turb // turbulent flow coefficient (R4)
expt // pressure exponent (R4)
rdens // reference fluid density [kg/m^3] {R4}
fdf // free delivery flow (prise = 0) [kg/s] {R4}
sop // shut-off pressure (flow = 0) [Pa] {R4}
off // fan is off if (RPM/rated RPM) < off {R4}
```

The second line consists of:

```
fpc[4] // fan performance polynomial coefficients {R4}
npts // number of mesaured data points (I2) {W}
Sarea // shut-off orifice area [m^2] {R4} {W}
u_Sa // units of shut-off area (I2) {W}
```

The next npts lines consists of:

```
mF // measured flow rates [kg/s] (R4) {W}
u_mF // units of measured flows (I2) {W}
dP // measured pressure rises [Pa] (R4) {W}
u_dP // units of pressure rises (I2) {W}
```

```
rP // revised pressure rises [Pa] (R4) {W}
u_rP // units of revised pressures (I2) {W}
```

Element type 30 [DD_BDQ] "dct_bdq" stored in structure DFE_BDQ.

The volume flow powerlaw backdraft damper data consist of:

```
lam // laminar flow coefficient {R4}
Cp // turbulent flow coefficient ( dP > 0 ) {R4}
xp // pressure exponent ( dP > 0 ) {R4}
Cn // turbulent flow coefficient ( dP < 0 ) {R4}
xn // pressure exponent ( dP < 0 ) {R4}
```

Element type 31 [DD_BDF] "dct_bdf" stored in structure DFE_BDF.

The mass flow powerlaw backdraft damper data consist of:

```
lam // laminar flow coefficient {R4}
Cp // turbulent flow coefficient ( dP > 0 ) {R4}
xp // pressure exponent ( dP > 0 ) {R4}
Cn // turbulent flow coefficient ( dP < 0 ) {R4}
xn // pressure exponent ( dP < 0 ) {R4}
```

After each element comes the following geometry and leakage data stored in structure DUCT.

The first line of duct geometry data consists of:

```
hdia // hydraulic diameter [m] {R4}
perim // perimeter [m] {R4}
area // cross sectional area [m^2] {R4}
major // major dimension of rectangular or oval duct [m] {R4}
minor // minor dimension of rectangular or oval duct [m] {R4}
As // duct segment surface area [m^2] {R4} (not used)
Qr // duct leakage rate at Pr [L/s/m^2] {R4}
Pr // dPstatic for leakage rate [Pa] {R4}
```

The second line of duct geometry data consists of:

```
shape // 0 = circle, 1 = rectangle, 2 = oval, 3 = other (I2) {W}
u_D // units for diameter (I2) {W}
u_P // units for perimeter (I2) {W}
u_A // units for area (I2) {W}
u_mj // units for major dimension (I2) {W}
u_mn // units for minor dimension (I2) {W}
u_Qr // units for leakage rate (I2) {W}
u_Pr // units for standard dP (I2) {W}
```

The duct element section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 12a: Control Super Elements

Control Super Element data are read by the ctrlse_read( ) function and saved by the ctrlse_save( ) function in ContamW. The data are stored in the CTRLSE_DAT structures that are defined in contam.h for ContamW. Super Element data are not read in by ContamX, only the nodes which

are instantiated from Control Super Elements are read in by ContamX as described in the following Section 12b.

The Super Elements section starts with:

```
_nselmt // number of super elements (IX)
```

This is followed by data for all _nselmt super elements.

For each super element the first data line includes:

```
nr // element number (IX); in order from 1 to _nselemt
flags   // if( flags & CF_DEF ) super element is defined (U2)
    // if( flags & CF_SNS ) super element is a sensor
in // input node number (IX)
out // output node number (IX)
nn // number of sub-nodes (IX)
ni // number of sketch icons (IX)
name[] // element name (I1)
```

and the second line has:

```
desc[] // element description (I1) may be blank
```

This is followed by a section that contains sub-node data of the Super Element. This will be very similar to the "control nodes:" section of the project file (see Section 12b below) and repeated within section 12b for each super node that is instantiated from a super element. However, the actual parameters of the instantiated super nodes may vary from those of the super element sub-nodes.

This is followed by lines of data that contain the SketchPad icon parameters of the Super Element sketch and will follow the "`levels plus icon data:`" section of the project file but only for a single "level" of data that represents the Super Element sub-node icons (see Section 3).

The Super Elements section is terminated with:

```
-999
```

Section 12b: Control Nodes

Control node data are read by the ctrl_read( ) function and saved by the ctrl_save( ) function. The data are stored in the CTRL_DSC structures that are defined in contam.h for ContamW and the CT_NODE structures defined simdat.h for ContamX.

The control nodes section starts with:

```
_nctrl // number of control nodes (IX); in order from 1 to _nctrl
```

This is followed by a data header comment line and then data for all _nctrl control nodes.

For each control node the first data line includes:

```
nr // node (SketchPad) number (IX); in order from 1 to _nctrl
type // node data type (string ? I2)
Node type names are stored in _ctrl_names in type order.
seqnr // computation sequence number (IX); set in ContamW
flags // flags for offset & scale, time constant, and 1D sensor (U2)
inreq // number of required inputs (I2) {W}
```

```
n1 // SketchPad number of input node #1 (IX)
n2 // SketchPad number of input node #2 (IX)
name[] // element name (I1) {W}
```

and the second line has:

```
desc[] // control node description (I1) {W} may be blank
```

This may be followed by one more line of data that depends on the node type.

Node type 0 [CT_SNS] "sns" stored in structure SENSOR {W} or SNSDAT {X}.

```
offset // offset value (R4)
scale // scale value (R4)
tau // time constant (R4)
oldsig // signal at last time step - for restart (R4)
source // index of source (source not defined at read time) (IX)
type // type of source: 1=zone, 2=path, 3=junction, 4=duct,
 // 5=exp, 6=term (I2)
measure // 0=contaminant, 1=temperature, 2=flow rate, 3=dP,
 // 4=Pgage, 5=zone occupancy (I2)
X // X-coordinate of sensor [m] (R4)
Y // Y-coordinate of sensor [m] (R4)
relHt // relative height of sensor [m] (R4)
units[] // units of coordinates {W} (I1)
species[] // species name [I1]; convert to pointer
```

Node type 1 [CT_SCH] "sch" stored in structure SCHDAT.

```
ps // week schedule index (I2); converted to pointer
```

Node type 2 [CT_SET] "set" stored in structure SETDAT.

```
value // constant value (R4)
```

Node type 3 [CT_CVF] "cvf" stored in structure CDVDAT. {Contam 2.1}

```
name[] // name of the value read from the Continuous Values file (I1)
```

Node type 4 [CT_DVF] "dvf" stored in structure CDVDAT. {Contam 2.1}

```
name[] // name of the value read from the Discrete Values file (I1)
```

Node type 5 [CT_LOG] "log" stored in structure LOGDAT.

```
offset // offset value (R4)
scale // scale value (R4)
udef // true if using default units (I2) {W}
header[] // header string (I1)
units[] // units string (I1)
```

Node type 6 [CT_PAS] "pas" has no additional data.

Node type 7 [CT_MOD] "mod" stored in structure MOD.

```
offset // offset value (R4)
scale // scale value (R4)
```

Node type 8 [CT_HYS] "hys" stored in structure HYSDAT.

```
slack; // hysteresis parameter (R4)
slope // 1.0 / (1.0 - slack) (R4)
oldsig // prior output signal (R4)
```

Node type 9 [CT_ABS] "abs" has no additional data.

Node type 10 [CT_BIN] "bin" has no additional data.

Node type 11 [CT_DLS] "dls" stored in structure CDVDAT. {Contam 2.1}

```
dsincr // day schedule number for increasing signal (I2)
dsdecr // day schedule number for decreasing signal (I2)
```

Node type 12 [CT_DLX] "dlx" stored in structure CDVDAT. {Contam 2.1}

```
tauincr // time constant for increasing signal [s] (I4)
taudecr // time constant for decreasing signal [s] (I4)
```

Node type 13 [CT_INT] "int" has no additional data.

Node type 14 [CT_RAV] "rav" stored in structure CDVDAT. {Contam 2.1}

```
tspan // time span for the running average [s] (I4)
```

Node type 15 [CT_INV] "inv" has no additional data.

Node type 16 [CT_AND] "and" has no additional data.

Node type 17 [CT_OR] "od" has no additional data.

Node type 18 [CT_XOR] "xor" has no additional data.

Node type 19 [CT_ADD] "add" has no additional data.

Node type 20 [CT_SUB] "sub" has no additional data.

Node type 21 [CT_MUL] "mul" has no additional data.

Node type 22 [CT_DIV] "div" has no additional data.

Node type 23 [CT_SUM] "sum" stored in structure SUMAVG.

Node type 24 [CT_AVG] "avg" stored in structure SUMAVG.

Node type 25 [CT_MAX] "max" stored in structure SUMAVG.

Node type 26 [CT_MIN] "min" stored in structure SUMAVG.

```
npcs // number of controls to be processed (IX)
pc[npcs] // indices of npcs control nodes (IX)
```

Node type 27 [CT_LLS] "lls" has no additional data.

Node type 28 [CT_ULS] "uls" has no additional data.

Node type 29 [CT_LBS] "lbs" stored in structure BANDAT.

Node type 30 [CT_UBS] "ubs" stored in structure BANDAT.

```
band // width of band (R4)
```

Node type 31 [CT_LLC] "llc" has no additional data.

Node type 32 [CT_ULC] "ulc" has no additional data.

Node type 33 [CT_PC1] "pc1" stored in structure PCDAT.

```
kp // proportional gain factor (R4)
```

Node type 34 [CT_PI1] "pi1" stored in structure PICDAT.

```
kp // proportional gain factor (R4)
```

```
ki // integral gain factor (R4)
oldsig // prior output signal - for restart (R4)
olderr // prior error value - for restart (R4)
```

Node type 35 [CT_SUP] "sup" stored in structure SUPDAT.

```
def // 0:super element undefined 1:defined (I2)
se // super element number (IX)
in // input sub-node number (IX)
out // output sub-node number (IX)
```

Node type 36 [CT_SEN] this is only a placeholder value for use with the type dialog.

Node type 37 [CT_SPH] "sph" has no additional data.

The control nodes section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 13: Simple Air Handling System (AHS)

AHS data are read by the system_read( ) function and saved by the system_save( ) function. The data are stored in the AHS_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The AHS section starts with:

```
_nahs // number of AHS (IX)
```

This is followed by a data header comment line and then data for all _nahs systems.

For each AHS the first data line includes:

```
nr // AHS number (IX); in order from 1 to _nahs
zone_r // return zone number (IX)
zone_s // supply zone number (IX)
path_r // recircultaion air path number (IX)
path_s // outdoor air path number (IX)
path_x // exhaust path number (IX)
```

and the second line has:

```
desc[] // AHS description (I1) {W} may be blank
```

The AHS section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 14: Zones

Zone data are read by the zone_read( ) function and saved by the zone_save( ) function. The data are stored in the ZONE_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The zone section starts with:

```
_nzone // number of zones (IX)
```

This is followed by a data header comment line and then data for all _nzone zones.

For each zone the data line includes:

```
nr // zone number (IX); in order from 1 to _nzone
flags // zone flags – bits defined in contam.h (U2)
```

```
ps // week schedule index (IX); converted to pointer
pc // control node index (IX); converted to pointer
pk // kinetic reaction index (IX); converted to pointer
pl // building level index (IX); converted to pointer
relHt // zone height [m] (R4)
Vol // zone volume [m^3] (R4)
T0 // initial zone temperature [K] (R4)
P0 // initial zone pressure [Pa] (R4)
name[] // zone name (I1) {W}
color // zone fill color (I2) {W} {Contam 2.4}
u_Ht // units of height (I2) {W}
u_V // units of volume (I2) {W}
u_T // units of temperature (I2) {W}
u_P // units of pressure (I2) {W}
cdaxis // conv/diff axis - (0=no cd, 1-4 => cd axis direction) (I2)
vf_type // 0=no value file, 1=use cvf, 2=use dvf (I2) {CONTAM 3.2}
vf_node_name[NAMELEN] // value file node name (I1) {CONTAM 3.2}
cfd // cfd zone (0=no, 1=yes) (I2) {CONTAM 3.0}
```

If cfd = 1 then the line continues to include: {CONTAM 3.0}

```
name[] // cfd zone id (I1)
```

If cdaxis > 0 then the line continues to include:

```
"1D:" // string indicating 1D data follows
X1        // X coordinate of one end of cdaxis
Y1        // Y     "            "
H1        // Relative Height    "
X2        // X coordinate of other end of cdaxis
Y2        // Y     "            "
H2        // Relative Height    "
celldx // length of c/d cell [m] (R4)
axialD // axial diffusion coeff [m^2/s] (R4)
u_aD // units of axial diffusion (I2)
u_L // units of c/d axis limits (I2)
```

The zones section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 15: Initial Zone Concentrations

Initial zone contaminant mass fractions are read by the zone_read( ) function and saved by the zone_save( ) function. The mass fractions are stored in the ZONE_DSC structures that are defined in contam.h for ContamW and they are stored in an array in ContamX.

The zone contaminants section starts with:

```
nn // number of mass fractions that follow (IX)
```

*nn should equal _nzone * _nctm.*

This is followed by _nzone lines in order from 1 to _nzone.

Each data line contains _nctm mass fractions (R4) in contaminant order.

```
nr   // zone number (IX)
```

```
CC0[i][1] // initial mass fraction (R4) of zone i, contaminant 1
...
CC0[i][_nctm] // initial mass fraction (R4) of zone i, contaminant _nctm
```

The initial zone concentrations section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 16: Airflow Paths

Path data are read by the path_read( ) function and saved by the path_save( ) function. The data are stored in the PATH_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The path section starts with:

```
_npath // number of paths (IX)
```

This is followed by a data header comment line and then data for all _npath paths.

For each path the data line includes:

```
nr // path number (IX); in order from 1 to _npath
flags // airflow path flag values (I2)
pzn // zone N index (IX); converted to pointer
pzm // zone M index (IX); converted to pointer
pe // flow element index (IX); converted to pointer
pf // filter index (IX); converted to pointer
pw // wind coefficients index (IX); converted to pointer
pa // AHS index (IX); converted to pointer
ps // schedule index (IX); converted to pointer
pc // control node index (IX); converted to pointer
pld // level index (IX); converted to pointer
X // X-coordinate of envelope path [m] (R4) {Contam 2.1}
Y // Y-coordinate of envelope path [m] (R4) {Contam 2.1}
relHt // height relative to current level [m] (R4)
mult // element multiplier (R4)
wPset // constant wind pressure [Pa] (pw==NULL) (R4)
wPmod // wind speed(?) modifier (pw!=NULL) (R4)
wazm // wall azimuth angle (pw!=NULL) (R4)
Fahs // AHS path flow rate [kg/s] (pw==NULL) (R4)
Xmax // flow or pressure limit - maximum (R4) {W}
Xmin // flow or pressure limit - minimum (R4) {W}
icon // icon used to represent flow path (U1) {W}
dir // positive flow direction on sketchpad (U1) {W}
u_Ht // units of height (I2) {W}
u_XY // units of X and Y (I2) {W}
u_dP // units of pressure difference (I2) {W}
u_F // units of flow (I2) {W}
vf_type // 0=no value file, 1=use cvf, 2=use dvf (I2) {CONTAM 3.2}
vf_node_name[NAMELEN] // value file node name (I1) {CONTAM 3.2}
cfd // cfd path (0=no, 1=yes) (I2) {CONTAM 3.0}
```

If cfd = 1 then the line continues to include: {CONTAM 3.0}

```
name[] // cfd path id (I1)
cfd_ptype // boundary condition type (0=mass flow, 1=pressure) (I2)
cfd_btype // pressure bc type (if ptype = 1, 0=linear, 1=stagnation
pressure) (I2)
cfd_capp // coupling approach (1=pressure-pressure) (I2)
```

The paths section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 17: Duct Junctions

Junction data are read by the jct_read( ) function and saved by the jct_save( ) function. The data are stored in the JCT_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The junction section starts with:

```
_njct // number of junctions (I2)
```

This is followed by a data header comment line and then data for all _njct junctions.

For each junction the data line includes:

```
nr // junction number (I2); in order from 1 to _njct
flags // junction flags - bits defined in contam.h (U2)
jtype // junction type flag: 0=jct, 1=term, 3=term w/ wind {Contam 2.1}
pzn // surrounding zone index (I2); converted to pointer
dnr // duct segment number - use during read (I2)
pk // kinetic reaction index (I2); converted to pointer
ps // schedule index (I2); converted to pointer
pc // control node index (I2); converted to pointer
pld // level index (I2); converted to pointer
X // X-coordinate of ambient terminal [m] (R4) {Contam 2.1}
Y // Y-coordinate of ambient terminal [m] (R4) {Contam 2.1}
relHt // height relative to current level [m] (R4)
T0 // initial junction temperature [K] (R4)
P0 // initial junction pressure [Pa] (R4)
icon // icon used to represent junction (U1) {W}
u_Ht // units of height (I2) {W}
u_XY // units of X and Y (I2) {W}
u_T // units of temperature (I2) {W}
u_dP // units of pressure difference (I2) {W}
vf_type // 0=no value file, 1=use cvf, 2=use dvf (I2) {CONTAM 3.2}
vf_node_name[NAMELEN] // value file node name (I1) {CONTAM 3.2}
```

If jtype > 0 then the line continues with:

```
"T:" // string indicates terminal data follows {Contam 2.3}
pf // filter index (I2); converted to pointer {Contam 2.3}
pw // wind coefficients index (I2); converted to pointer
wPset // constant wind pressure [Pa] (pw==NULL) (R4)
wPmod // wind speed(?) modifier (pw!=NULL) (R4)
wazm // wall azimuth angle (pw!=NULL) (R4)
bal // balance terminal flag (I2) {Contam 2.3}
Ad // area of duct connected to terminal [m^2] (R4) {Contam 2.3}
```

```
Af // free area of terminal [m^2] (R4) {Contam 2.3}
Fdes // design flow rate [kg/s] (R4) {Contam 2.3}
Ct // terminal loss coefficient (R4) {Contam 2.3}
Cb // balance loss coefficient (R4) {Contam 2.3}
Cbmax // maximum Cb desired (R4) {Contam 2.3}
u_A  // units of area (I2) {Contam 2.3}
u_F // units of mass flow (I2) {Contam 2.3}
ddir // direction of terminal duct - to show wind pressure (U1) {W}
fdir // positive flow direction - to show flow (U1) {W}
```

The junctions section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 18: Initial Junction Concentrations

Initial junction contaminant mass fractions are read by the zone_read( ) function and saved by the zone_save( ) function. The mass fractions are stored in the ZONE_DSC structures that are defined in contam.h for ContamW and they are stored in an array in ContamX.

The initial junction contaminants section starts with:

```
nn // number of mass fractions that follow (IX)
```

*nn should equal _njct * _nctm. This is followed by _njct lines in order from junction 1 to junction _njct.*

Each data line contains _nctm mass fractions (R4) in contaminant order.

The initial junction concentrations section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 19: Duct Segments

Duct data are read by the duct_read( ) function and saved by the duct_save( ) function. The data are stored in the DCT_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The duct segment section starts with:

```
_ndct // number of ducts (IX)
```

This is followed by a data header comment line and then data for all _ndct ducts.

For each duct the data line includes:

```
nr // duct number (IX); in order from 1 to _ndct
flags // duct flag values (I2)
pjn // junction n index (IX); converted to pointer
pjm // junction m index (IX); converted to pointer
pe // duct flow element index (IX); converted to pointer
pf // filter index (IX); converted to pointer
ps // schedule index (IX); converted to pointer
pc // control node index (IX); converted to pointer
dir // positive flow direction on sketchpad (U1) {W}
length // length of the duct segment [m] (R4)
Ain // flow area at inlet end [m^2] - future (R4)
Aout // flow area at outlet end [m^2] - future (R4)
```

```
sllc // sum of local loss coefficients (R4)
color // icon color (I2) {W} {Contam 2.4}
u_L // units for length (I2) {W}
u_A // units for flow area (I2) {W}
vf_type // 0=no value file, 1=use cvf, 2=use dvf (I2) {CONTAM 3.2}
vf_node_name[NAMELEN] // value file node name (I1) {CONTAM 3.2}
```

The duct segment section is terminated with:

```
-999
```

Section 20: Source/Sinks

Contaminant sources/sinks are read by the css_read( ) function and saved by the css_save( ) function. The data are stored in the CSS_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The source/sink section starts with:

```
_ncss // number of source/sinks (IX)
```

This is followed by a data header comment line and then data for all _ncss source/sinks.

For each source/sink the data line includes:

```
  nr // source/sink number (IX); in order from 1 to _ncss
  pz // zone index (IX); converted to pointer
  pe // source/sink element index (IX); converted to pointer
  ps // schedule index (IX); converted to pointer
  pc // control node index (IX); converted to pointer
  mult // multiplier (R4)
  CC0 // initial mass fraction (some types) [kg/kg] (R4)
  Xmin // minimum x coordinate of source/sink location [m] (R4)
  Ymin // minimum y coordinate of source/sink location [m] (R4)
  Hmin // minimum z coordinate relative to level [m] (R4)
  Xmax // maximum x coordinate of source/sink location [m] (R4)
  Ymax // maximum y coordinate of source/sink location [m] (R4)
  Hmax // maximum z coordinate relative to level [m] (R4)
  u_XYZ // units of location coordinates (I2)
  vf_type // 0=no value file, 1=use cvf, 2=use dvf (I2) {CONTAM 3.2}
  vf_node_name[NAMELEN] // value file node name (I1) {CONTAM 3.2}
  cfd // cfd path (0=no, 1=yes) (I2) {CONTAM 3.0}
```

If cfd = 1 then the line continues to include: {CONTAM 3.0}

```
  name[] // cfd path id (I1)
```

The source/sink section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 21: Occupancy Schedules

Occupancy schedules are read by the oschd_read( ) function and saved by the oschd_save( ) function. The data are stored in the OD_SCHD structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The occupancy schedule section starts with:

```
_nosch // number of schedules (IX)
```

This is followed by a data header comment line and then data for all _nosch schedules.

For each schedule the first data line includes:

```
nr // schedule number (IX); in order from 1 to _nosch
npts // number of points (I2)
u_XYZ // units of location coordinates (I2)
name[] // schedule name (I1) {W}
```

and the second line has:

```
desc[] // schedule description (I1) {W} may be blank
```

This is followed by npts lines of schedule data:

```
time // time-of-day [s] (hh:mm:ss converted to I4)
pz // zone index (I2); converted to pointer
X // X-coordinate of occupant [m] (R4)
Y // Y-coordinate of occupant [m] (R4)
relHt // height relative to current level [m] (R4)
```

The occupancy schedule section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 22: Exposures

Exposure data are read by the pexp_read( ) function and saved by the pexp_save( ) function. The data are stored in the PEXP_DSC structures that are defined in contam.h for ContamW and simdat.h for ContamX.

The exposure section starts with:

```
_npexp // number of exposures (IX)
```

This is followed by three lines of data for all _npexp exposures.

For each exposure the first data line includes:

```
nr // exposure number (IX); in order from 1 to _npexp
gen // = 1 if contaminants are generated (I2)
ncg // number of contaminant generations (I2)
cgmlt // contaminant generation multiplier [-] (R4)
```

NOTE - the following 5 items were removed from the project file for version 3.2:

```
inhmax // peak inhalation rate [m^3/s] (R4) {unused}
inhsch // weekly inhalation schedule index (I2)
bodywt // body weight [kg] (R4) {unused}
u_inh // units for inhalation rate (I2)
u_bw // units for body weight (I2)
```

The second line has the exposure description:

```
desc[] // exposure/person description (I1)
```

The third line has the indices of 12 occupancy schedules:

```
odsch[12] // vector of daily occupancy schedules - 12 indices
```

It is followed by ncg lines of contaminant generation data:

```
name // species name (I1)
```

```
ps // schedule index (I2); converted to pointer
cgmax // peak generation rate [kg/s] (R4)
u_cg // units of generation rate (I2) {W}
vf_type // 0=no value file, 1=use cvf, 2=use dvf (I2) {CONTAM 3.2}
```

If `vf_type` > 0 then the line continues to include:

```
vf_node_name[NAMELEN] // value file node name (I1) {CONTAM 3.2}
```

The exposure section is terminated with:

```
-999 // used to check for a read error in the above data
```

Section 23: Annotations

Annotations are read by the note_read( ) function and saved by the note_save( ) function. The data are stored in the NOTE_DSC structures that are defined in contam.h. Annotations are used only in ContamW; nothing is stored when this section is read in ContamX.

The annotations section starts with:

```
_nnote // number of annotations (IX)
```

This is followed by data for all _nnote annotations:

For each annotation data line includes:

```
nr // annotation number (IX) {W}; in order from 1 to _nnode
note[] // annotation (I1) {W}
```

The annotations section is terminated with:

```
-999 // used to check for a read error in the above data
```

APPENDIX B - CONTAMX TCP/IP SOCKET COMMUNICATION

The socket communication features of ContamX enable another application to control the execution of ContamX as it is running. During this *bridge* mode of execution, ContamX is referred to as the client, and the controlling program as the server. These two programs communicate with each other using the TCP/IP Socket communication method and a pre-defined set of messages. Generally, the server must be developed by the user. New socket communications messages have been added, and ContamX has been modified to enable sending and receiving a wider set of ContamX-related information including zone temperatures and detailed airflow results.

The ability to couple ContamX with both TRNSYS and EnergyPlus has also been developed. A new TRNSYS Type98 has been developed as a means to demonstrate these new capabilities (see Working with TRNSYS and Working with EnergyPlus).

This section provides the details of executing ContamX in the socket mode and the specifications of the pre-defined set of messages implemented by ContamX. These messages fall into the following categories defined from the perspective of ContamX:

- Send *project information*
- Send/Receive *execution control*
- Receive *project modification*
- Send *simulation results*

Program Execution

In order to activate the socket communication capabilities of ContamX, the program must be run with the following command line structure.

```
contamx3 <project> -b [-wp][-vf] <address> : <port>

contamx3 program executable
<project> name of the CONTAM project file
-b Run ContamX in bridge mode (required)
options:
-wp ContamX calculates wind pressure using traditional methods as determined
by the PRJ file (none, constant, or variable). Wind speed and direction for
variable pressure flow paths should be provided to ContamX via the
ADJ_WTH_MSGTYPE message. If this command line option is not used then wind
pressure and ambient contaminants must be provided via the ADJ_WPC_MSGTYPE
message.
-vf contamx socket messages provide airflow rates in volume flow units [m³/s]
as opposed to the default [kg/s].
<address> ip address of process (server) that will be controlling this
instance of ContamX (client)
<port> port number of address through which socket is established. If
omitted, a default of 8200 will be assumed.
```

After ContamX starts, it will attempt to make contact with the controlling application and establish a socket connection. Once a connection has been established, ContamX will send the *project information* messages (types 1 – 13) to the server to provide information on the contents of the project file. These project information messages will be sent in numerical order. ContamX will then perform the steady state initialization calculation and send all of the *simulation result* messages (`_UPDATE_MSGTYPE`) followed by the `CX_READY_MSGTYPE`. At this point the server can begin interacting with ContamX in the bridge mode via the *execution control* and *project modification* messages detailed below. ContamX will respond with the requested `_UPDATE_MSGTYPE` messages and indicate when it is ready to receive more messages by sending the `CX_READY_MSGTYPE` message.

NOTE: Set the `_echo` value in the PRJ file greater than 1 to have bridge message information written to the .log file. This can provide insight into bridge execution behavior.

ContamX TCP/IP Message Formats

This section provides the detailed messages and formats. There are currently four categories of messages: *Project information*, *Execution control*, *Project modification* and *Simulation result*. *Project information* messages provide information related to the PRJ file in order for a server to establish the type and number of building components contained in a CONTAM project. These messages will all be sent by ContamX immediately upon execution of ContamX in the bridge mode. *Execution control* messages provide the ability to advance the simulation to selected simulation times until simulation is complete, to notify the server that ContamX is ready to receive messages, and to terminate the simulation. *Project modification* messages are those received by ContamX in order to adjust various properties of the building during simulation including zone contaminant concentrations, temperatures of zones and duct junctions, control node values, weather data and other items detailed below. *Simulation result* messages are sent by ContamX to provide the results of the simulation for the current time step.

In the following messages, any value of type `int` will be converted to network byte order before being packaged into the message, but values of type `char` and `float` will not. Server software written to work with ContamX will need to manage conversion of data as needed depending on the computer architecture/operating systems involved, but this should not be an issue if both the server and ContamX are running on the same computer.

Message Header and Message IDs

The message header is the first part of every message. It provides a consistent means of establishing the specific information that is to follow. The specifics will depend on the `type` field of the message. The `length` field includes the length of the header and the specific portion of the message that follows. The `version` field is provided to enable programs that utilize the socket communication capabilities of ContamX to ensure conformity to the messaging requirements as they evolve due to modifications/enhancements of the methodology.

```
char pattern[8];  // string to verify the header 'ACATMSG'.
int length; // length of entire message in bytes
int type; // type of message (see table below.)
int version; // version # = 9
```

Message Type Definitions

Project information messages:

```
BLDG_INFO_MSGTYPE 1
SIMPARM_INFO_MSGTYPE 2
AGENT_INFO_MSGTYPE 3
ELEMENT_INFO_MSGTYPE 4
INPUT_CTRL_INFO_MSGTYPE 5
OUTPUT_CTRL_INFO_MSGTYPE 6
AHSP_INFO_MSGTYPE 7
ZONE_INFO_MSGTYPE 8
PATH_INFO_MSGTYPE 9
JCT_INFO_MSGTYPE 10
TERM_INFO_MSGTYPE 11
LEAK_INFO_MSGTYPE 12
DUCT_INFO_MSGTYPE 13
```

Execution control messages:

```
CX_READY_MSGTYPE  0
CX_ADVANCE_MSGTYPE  20
CX_ERROR_MSGTYPE 200
```

Project modification messages:

```
ADJ_ZONE_CONC_MSGTYPE  30
ADJ_ZONE_TEMP_MSGTYPE  40
ADJ_JCT_TEMP_MSGTYPE  50
ADJ_ELEMENT_MSGTYPE  60
ADJ_CONTROL_NODE_MSGTYPE  70
ADJ_WTH_MSGTYPE  80
ADJ_WPC_MSGTYPE  90
```

Simulation result messages:

```
CONC_UPDATE_MSGTYPE  100
PATH_FLOW_UPDATE_MSGTYPE  110
TERM_FLOW_UPDATE_MSGTYPE  120
AHSP_FLOW_UPDATE_MSGTYPE  130
DUCT_FLOW_UPDATE_MSGTYPE  140
LEAK_FLOW_UPDATE_MSGTYPE  150
CTRL_NODE_UPDATE_MSGTYPE  160
```

BLDG_INFO_MSGTYPE

Provides the name of the CONTAM project file.

```
Message Header;
char* building_name; // CONTAM project file name without .PRJ extension
```

SIMPARM_INFO_MSGTYPE

Provides simulation control parameters.

```
Message Header;
int date0; // simulation start date
int date1; // simulation end date
int time0; // simulation start time
int time1; // simulation end time
int timestep; // simulation time step [s]
```

AGENT_INFO_MSGTYPE

Provides information related to the contaminants, i.e., only those species that are being used in the simulation.

```
Message Header;
int num_agents; // number of contaminants in the PRJ file
int id[num_agents]; // index of contaminants according to ContamX
char *name[num_agents]; // contaminant names (null-terminated strings)
```

ELEMENT_INFO_MSGTYPE

Provides a list of airflow elements in the order in which they occur in the project file. This provides the ability to change the airflow element, via the ADJ_ELEMENT_MSGTYPE, to which an airflow path refers using the index of the element in the list of names provided by this message.

```
Message Header;
int num_elements; // number of airflow elements (not ducts) in the PRJ
char element_names[num_elements]; // name of each element (null-terminated
strings)
```

INPUT_CTRL_INFO_MSGTYPE

Provides number of control input nodes and the initial value of each node at the start of the simulation. Control input nodes are defined via ContamW by providing *Constant* type control nodes with a name. The values of these control nodes can be modified during simulation via the ADJ_CTRL_NODE_MSGTYPE.

```
Message Header;
int num_nodes; // number of control input nodes in the PRJ file
int id[num_nodes]; // index of control nodes according to ContamX
float  init_vals[num_nodes]; // initial values of control input nodes
char* name[num_nodes]; // input node names (null-terminated strings)
```

OUTPUT_CTRL_INFO_MSGTYPE

Provides initial values of control output nodes at the start of the simulation. Control output nodes are defined via ContamW by providing *Signal Split* type control nodes with a name. Values can be obtained during simulation via the `CTRL_NODE_UPDATE_MSGTYPE`.

```
Message Header;
int num_nodes; // number of control output nodes in the PRJ file
int id[num_nodes]; // index of control nodes according to ContamX
float  init_vals[num_nodes]; // initial values of control input nodes
char* name[num_nodes]; // output node names (null-terminated strings)
```

AHSP_INFO_MSGTYPE

Provides information related to simple air handling system airflow paths. AHS paths include return, supply and implicit airflow paths (recirculation, outdoor air and exhaust) of simple air handling systems. Supply and return airflow paths will come before all of the implicit airflow paths which will come in groups of three: recirculation, outdoor air and exhaust flow paths for each air handler. This is apparent when viewing the ContamW airflow plotting dialog box. The `id[]` array provides the order in which airflows will be provided by ContamX via the `AHSP_FLOW_UPDATE_MSGTYPE`. If the `ambient_index` is 0, the path is not a building envelope path. Otherwise, the ambient_index specifies the order in which path concentrations are to be sent using the `ADJ_WPC_MSGTYPE`. Only implicit outdoor air intake paths can be considered envelope paths.

```
Message Header;
int num_ahsp; // number of AHS airflow paths in the PRJ file
int id[num_ahsp]; // index of each AHS path according to ContamX
int ambient_flag[num_paths] // 0 = not connected to ambient
float x[num_paths]; // x coordinate [m]
float y[num_paths]; // y coordinate [m]
float z[num_paths]; // if ambient_index > 0 then absolute height, else
relative to level [m]
```

ZONE_INFO_MSGTYPE

Provides zone information in the order in which concentrations will be sent back by ContamX via the `CONC_UPDATE_MSGTYPE`. This list of zones includes those of the two implicit zones (supply and return) of each simple air handling system.

```
Message Header;
int num_zones; // number of zones in the PRJ file
int level[num_zones]; // level number on which zone is located
int ahs[num_zones]; // 0 = not simple ahs zone, 1 = implicit ahs zone
int volume[num_zones]; // volume of each zone [m3]
char *name[num_zones]; // zone names (null-terminated strings)
```

PATH_INFO_MSGTYPE

Provides information related to "standard" airflow paths (as opposed to AHS paths). The `id[]` array provides the order in which airflow rates will be provided by ContamX via the `PATH_FLOW_UPDATE_MSGTYPE`. If the `ambient_index` is 0, the path is not an envelope infiltration point. Otherwise, the `ambient_index` specifies the order in which path

concentrations are to be sent using the ADJ_WPC_MSGTYPE. Positive flow direction is based on airflow path data to be from zone n to zone m.

```
Message Header;
int num_paths; // number of airflow paths in the PRJ
int id[num_paths]; // index according to ContamX
int ambient_index[num_paths]  // 0 = not connected to ambient
int element_index[num_paths]; // index of initial airflow element
int from_zone[num_paths]; // zone number of from zone (based on positive flow
direction)
int to_zone[num_paths];   // zone number of to zone
float x[num_paths]; // x coordinate [m]
float y[num_paths]; // y coordinate [m]
float z[num_paths]; // if ambient_index > 0 then absolute height, else
relative to level [m]
```

JCT_INFO_MSGTYPE

Provides the number of junctions and the type of each junction. The index of the type[] array is used to set the temperature of the junctions via the ADJ_JCT_TEMP_MSGTYPE.

```
Message Header;
int num_jcts; // number of junctions in the PRJ file
int type[num_jcts]; // junction type: 0=junction, 1=terminal
```

TERM_INFO_MSGTYPE

Provides information on those junctions that are terminal types. The id[] array provides the order in which airflow rates will be provided by ContamX via TERM_FLOW_UPDATE_MSGTYPE. If the ambient_index is 0, this terminal is not an infiltration point. Otherwise, ambient_index specifies the order in which terminal concentrations are to be sent to ContamX using ADJ_WPC_MSGTYPE.

```
Message Header;
int num_terms; // number of duct terminals in the PRJ
int id[num_terms]; // for each terminal: index according to ContamX
int ambient_index[num_terms] // for each terminal: 0/1 = not connected /
connected to ambient
float x[num_terms]; // for each terminal: absolute coordinates [m]
float y[num_terms];
float z[num_terms];
```

LEAK_INFO_MSGTYPE

Provides a list of only those duct junctions which have been defined via ContamW to have leakage. The id[] array provides the order in which airflow rates will be provided by ContamX via LEAK_FLOW_UPDATE_MSGTYPE. Currently leaks are not included when using ADJ_WPC_MSGTYPE., so ambient_index will be zero for all leaks.

```
Message Header;
int num_leaks;     // number of junctions with leaks in the PRJ
int id[num_leaks]; // index according to ContamX
```

DUCT_INFO_MSGTYPE

Provides the number of duct segments that will correspond to the number of and the order in which duct segment airflow rates will be sent by ContamX via `DUCT_FLOW_UPDATE_MSGTYPE`.

```
Message Header;
int num_ducts;      // number of duct segments in the PRJ file
int id[num_ducts]; // for each duct segment: index according to ContamX
```

CX_READY_MSGTYPE

This message will be sent by ContamX to indicate that it is in the *ready mode* and is prepared to receive messages through the bridge. This will occur for the first time after ContamX has finished sending all of the *project information messages*, performed the steady state initialization calculation and sent all of the *simulation result messages* (`_UPDATE_MSGTYPE`). It will be sent again after each `CX_ADVANCE_MSGTYPE` has been handled and the *simulation result messages* have been sent as requested via the `option` flag of `CX_ADVANCE_MSGTYPE`.

```
Message Header;
int time; // time of day in seconds at which ContamX performed its most
recent calculation
```

CX_ADVANCE_MSGTYPE

This message is received by ContamX at any time while in the *ready mode* to notify ContamX to run until the simulation time specified in seconds from time 0:00:00 of the first day of the simulation as defined by the simulation start time in the PRJ file. For example, if the simulation begins at 09:30:00 and the simulation time step is 5 minutes, then the first `time` sent should be at least 34200 s + 1800 s. The `option` field is used to indicate the data ContamX should send back in the form of *Update Messages* after it reaches the time specified.

```
Message Header;
int option; // bitwise flag controls which _UPDATE_MSGTYPES ContamX will send
            // once time has been reached
  o  bit 1: iterative advance message (currently not used)
  o  bit 2: send CONC_UPDATE_MSGTYPE
  o  bit 3: send PATH_UPDATE_MSGTYPE
  o  bit 4: send AHSP_UPDATE_MSGTYPE
  o  bit 5: send DUCT_UPDATE_MSGTYPE
  o  bit 6: send TERM_UPDATE_MSGTYPE
  o  bit 7: send LEAK_UPDATE_MSGTYPE
  o  bit 8: send CTRL_NODE_UPDATE_MSGTYPE
int time; // simulation time in seconds to which ContamX should advance
relative to
          // time 0:00:00 of the first day of the simulation
```

CX_ERROR_MSGTYPE

This message may be received by ContamX at any time while in the **ready mode**, and will cause ContamX to terminate the simulation and shut down. This message is not required to terminate the program if it is allowed to run until the end of a simulation as determined by the simulation stop date/time in the PRJ file. If `error_type` sent is 0, then ContamX terminates immediately with an error. If `error_type` sent is 1, then ContamX will run until the end of the current day. This is used when coupling with EnergyPlus to accommodate warm-up days.

This message is sent by ContamX to the server to signal that it is shutting down. If ContamX shuts down with an error, it will send the message with the `error_type` parameter set to 1, otherwise it will be set to 0.

```
Message Header;
char error_type;
```

ADJ_ZONE_CONC_MSGTYPE

This message may be received by ContamX at any time while in the *ready mode*. Contaminant concentrations or masses should be sent in order of zones specified in the list of `zone_ids`. Each message can be used to set the concentration/mass of multiple zones, but when dealing with multiple agents, a separate message must be sent for each agent. The zone and agent ids used here are those provided by `AGENT_INFO_MSGTYPE` and `ZONE_INFO_MSGTYPE` respectively.

```
Message Header;
int option;     // 0 = add mass [kg], 1 = set concentration [kg/kg]
int num_zones; // number of zones to set
int agent_id;  // contaminant index
int zone_ids[num_zones]; // list of zone indexes
float values[num_zones]; // list of masses -or- concentrations
```

ADJ_ZONE_TEMP_MSGTYPE

This message may be received by ContamX at any time while in the *ready mode*. Each message can be used to set the temperature of multiple zones. The IDs used here are those provided by `ZONE_INFO_MSGTYPE`.

```
Message Header;
int num_zones;          // number of zones to set
int id[num_zones];      // list of zone indexes
float value[num_zones]; // temperatures [K]
```

ADJ_ELEMENT_MSGTYPE

This message can be received by ContamX at any time while in the *ready mode*. It is used to change the airflow element of an airflow path from one element to another. This could be useful, for example, to change a closed door element to a two-way open door element.

```
Message Header;
int path_id; // id of air flow path
int element_index; // index of the new element the path is to reference
```

ADJ_CONTROL_NODE_MSGTYPE

This message can be received by ContamX at any time while in the *ready mode*. It is used to set the value of a *Constant* (*Set* type, CT_SET) control node where `node_id` is the identifier assigned to the control node by ContamX and provided by `INPUT_CTRL_INFO_MSGTYPE`. Note that the `node_id` will not likely correspond to the control node number shown on the ContamW SketchPad for the corresponding control icon. However, node names are provided by `INPUT_CTRL_INFO_MSGTYPE`.

```
Message Header;
int node_id; // id of the control node
float value; // value to which node should be changed
```

ADJ_WTH_MSGTYPE

This message may be received by ContamX at any time while in the *ready mode*. Each message can be used to set the weather conditions which include temperature, barometric pressure, wind speed, and wind direction akin to the manner in which the WTH file is used as opposed to the detailed wind pressure information available via the WPC file and `ADJ_WPC_MSGTYPE`. See the Program Execution section for information related to the use of this message.

```
Message Header;
float temperature;       // ambient dry bulb temperature [K]
float pressure;             // atmospheric pressure at PRJ reference height
[Pa]
float wind speed;         // wind speed [m/s]
float wind direction; // direction from which wind is blowing [degrees]
```

ADJ_WPC_MSGTYPE

This message can be received by ContamX at any time while in the *ready mode*. It may be used to set either the ambient concentrations or pressures at envelope airflow paths including outdoor air intakes of simple air handling systems and duct terminals. Concentrations for paths, simple air handling system outdoor air intakes and terminals are transmitted in the order specified by the ambient indexes provided by the `PATH_INFO_MSGTYPE`, `AHSP_INFO_MSGTYPE`, and `TERM_INFO_MSGTYPE` messages. If multiple agents need to be transmitted, the order is:

```
 [all agents for path 1],
 ...
 [all agents for path num_paths],
 [all agents for ahs path 1],
  ...
 [all agents for path num_ahsp],
 [all agents for terminal 1],
 ...
 [all agents for terminal num_terms]
```

If `num_agents` is 0, then instead of concentrations, the message should provide one set of pressures corresponding to the pressures at the envelope locations. Pressures must conform to the 2.4 format of the WPC file as specified in the WPC File Format section of the documentation.

```
Message Header;
int time; // (value required but currently not used)
```

```
int num_agents; // number of contaminants OR set to zero to send pressures
int num_vals; // number of items to send depending on num_agents parameter
```

For contaminants, i.e., num_agents > 0:

```
int agent_ids[num_agents]; // index of each contaminant for which values are being sent
float values[num_agents * nEnvPaths]; // concentrations [kg_cont/kg_air]
```

For wind pressures, i.e., num_agents == 0:

```
float values[nEnvPaths]; // wind pressures [Pa]
```

CONC_UPDATE_MSGTYPE

This message may be sent by ContamX after `CX_ADVANCE_MSGTYPE` is received and ContamX has advanced to the time indicated. ContamX will only send this message if the associated bit of the `option` flag of the previous `CX_ADVANCE_MSGTYPE` is set. Values are sent in the order provided by `ZONE_INFO_MSGTYPE`.

```
Message Header;
int time;     // simulation time to which data corresponds [s]
int agent_id; // contaminant index of agent being updated
float concentrations[num_zones]; // contaminant concentrations [kg/kg]
```

PATH_FLOW_UPDATE_MSGTYPE

This message may be sent by ContamX after `CX_ADVANCE_MSGTYPE` is received and ContamX has advanced to the time indicated. ContamX will only send this message if the associated bit of the `option` flag of the previous `CX_ADVANCE_MSGTYPE` is set. Each path has two flow rates associated with it (flow0, flow1) to handle two-way airflow elements. flow0 and flow1 are sent consecutively for each path in the order provided by `PATH_INFO_MSGTYPE`. See the Program Execution section for information related to the units utilized by this message.

```
Message Header;
int time; // simulation time to which data corresponds [s]
float value[num_paths * 2]; // airflow rate for each airflow path [kg/s] or [m3/s]
```

TERM_FLOW_UPDATE_MSGTYPE

This message may be sent by ContamX after `CX_ADVANCE_MSGTYPE` is received and ContamX has advanced to the time indicated. ContamX will only send this message if the associated bit of the `option` flag of the previous `CX_ADVANCE_MSGTYPE` is set. Values are sent in the order provided by `TERM_INFO_MSGTYPE`. See the Program Execution section for information related to the units utilized by this message.

```
Message Header;
int time; // simulation time to which data corresponds [s]
float value[num_terms]; // airflow rate for each duct terminal [kg/s] or [m3/s]
```

AHSP_FLOW_UPDATE_MSGTYPE

This message may be sent by ContamX after CX_ADVANCE_MSGTYPE is received and ContamX has advanced to the time indicated. ContamX will only send this message if the associated bit of the option flag of the previous CX_ADVANCE_MSGTYPE is set. Values are sent in the order provided by AHSP_INFO_MSGTYPE. See the Program Execution section for information related to the units utilized by this message.

```
Message Header;
int time; // simulation time to which data corresponds [s]
float value[num_ahsp]; // airflow rate for each simple air handler path
[kg/s] or [m3/s]
```

DUCT_FLOW_UPDATE_MSGTYPE

This message may be sent by ContamX after CX_ADVANCE_MSGTYPE is received and ContamX has advanced to the time indicated. ContamX will only send this message if the associated bit of the option flag of the previous CX_ADVANCE_MSGTYPE is set. Values are sent in the order provided by DUCT_INFO_MSGTYPE. See the Program Execution section for information related to the units utilized by this message.

```
Message Header;
int time; // simulation time to which data corresponds [s]
float values[num_ducts]; // airflow rate for each duct segment [kg/s] or
[m3/s]
```

LEAK_FLOW_UPDATE_MSGTYPE

This message may be sent by ContamX after CX_ADVANCE_MSGTYPE is received and ContamX has advanced to the time indicated. ContamX will only send this message if the associated bit of the option flag of the previous CX_ADVANCE_MSGTYPE is set. Values are sent in the order provided by LEAK_INFO_MSGTYPE. See the Program Execution section for information related to the units utilized by this message.

```
Message Header;
int time; // simulation time to which data corresponds [s]
float values[num_leaks]; // airflow rate for each duct leak [kg/s] or [m3/s]
```

CTRL_NODE_UPDATE_MSGTYPE

This message may be sent by ContamX after CX_ADVANCE_MSGTYPE is received and ContamX has advanced to the time indicated. ContamX will only send this message if the associated bit of the option flag of the previous CX_ADVANCE_MSGTYPE is set. Values are sent in the order provided by OUTPUT_CTRL_INFO_MSGTYPE. Units depend on the control network.

```
Message Header;
int time; // simulation time to which data corresponds [s]
float values[num_outnodes]; // value of each control output node [-]
```