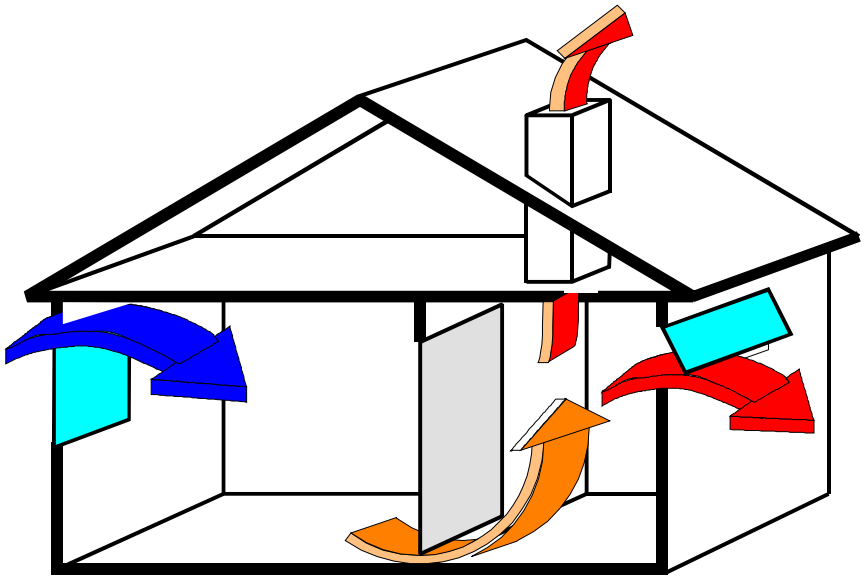
COMIS 3.1 - User's Guide

Edited by: Helmut E. Feustel and Brian V. Smith, LBNL

Revised edition by Viktor Dorer, Anne Haas and Andreas Weber, EMPA

January 24, 2001



Eidgenössische Materialprüfungs- und Forschungsanstalt
Laboratoire fédéral d'essai des matériaux et de recherche
Laboratorio federale di prova dei materiali e di ricerca
Institut federal da controlla da material e da retschertgas
Swiss Federal Laboratories for Materials Testing and Research

EMPA
Überlandstrasse 129
CH-8600 Dübendorf
Tel. +41-1-823 55 11
Fax +41-1-821 62 44

EMPA
Energiesysteme / Haustechnik
Energierechengruppe

PREFACE

The COMIS workshop (Conjunction of Multizone Infiltration Specialists) was a joint research effort to develop a multizone infiltration model. This workshop (October 1988 - September 1989) was hosted by the Energy Performance of Buildings Group at Lawrence Berkeley Laboratory's Applied Science Division. The task of the workshop was to develop a detailed multizone infiltration program taking crack flow, HVAC-systems, single-sided ventilation and transport mechanism through large openings into account. The agenda integrated all participants' contributions into a single model containing a library of modules. The user-friendly program is aimed at researchers and building professionals.

The work was accomplished not by investigating numerical description of physical phenomena but by reviewing the literature for the best suitable algorithm. The numerical description of physical phenomena clearly was a task of IEA-Annex 20 "Air Flow Patterns in Buildings" which finished in September 1991. In Annex 23 "Multizone Air Flow Modeling", which was adopted by the IEA-Energy Conservation in Buildings and Community Systems program in 1992, COMIS has been evaluated by means of tracer gas measurements, wind tunnel data, intermodel comparison, and comparison with analytical solutions.

From the time of its announcement in December 1986 COMIS was well received by the research community. Due to the internationality of the group, several national and international research programs were co-ordinated with the COMIS workshop. Colleagues from France, Greece, Italy, Japan, The Netherlands, People's Republic of China, Spain, Sweden, Switzerland, and the United States of America were working together on the development of the model and its evaluation.

Even though this kind of co-operation is well known in other fields of research, e.g., high energy physics, in the field of building physics it is a new approach.

The COMIS User's Guide contains an overview of the COMIS project as well as hints which will be useful in getting through the input and calculation procedure. The handbook comes in loose-leaf form so as to be easily updated according to the progress of the model development. Please note, that the COMIS User's Guide reflects the construction of the input file needed to run the calculation program COMVEN. There are several user interfaces available to create the input file and to run the program. Each interface comes with its own User's Guide.

Helmut E. Feustel
COMIS Co-ordinator and Annex 23 Operating Agent
Berkeley, California
August 31, 1997

Preface to version 3.1

This present version of the COMIS User Guide is the result of a major revision undertaken by EMPA. The update has gone together with an effort to debug and streamline the COMIS code, and to make code and User Guide as consistent as possible.

The changes from COMIS version 3.0 are summarized in chapter 1.4

This task has been supported by the help and the valuable contributions of:
Peter G. Schild (NBI, Norway), Brian Smith and Dave Lorenzetti (LBNL, USA), Hans C. Phaff (TNO, The Netherlands), Werner Keilholz (CSTB, France).

Viktor Dorer, Anne Haas and Andreas Weber
EMPA Duebendorf, Switzerland
January 24, 2001

Contents

1 Introduction **9**

1.1 The COMIS Project . . . 10

1.2 The original COMIS Model (COMIS Version 1.0) . . . 10

Input . . . 10

Flow through Building Components . . . 10

Solver . . . 12

Wind Pressures . . . 12

Products of the COMIS workshop . . . 12

1.3 Follow-Up of COMIS: IEA Annex 23 . . . 13

Products of IEA Annex 23 . . . 13

1.4 COMIS version 3.1 . . . 14

2 Getting Started **17**

2.1 Nomenclature in COMIS User Guide . . . 18

2.2 Distribution . . . 19

2.3 Installation . . . 19

Installation on a PC . . . 19

Compilation of COMIS . . . 19

2.4 COMIS File Structure (Stand-alone installation) . . . 20

2.5 COMIS User Interfaces . . . 22

COMERL SHELL User Interface . . . 22

IISiBat User Interface . . . 22

COMISExcel User Interface . . . 22

2.6 COMIS Tools . . . 23

3 Basic Concepts **25**

3.1 Input Data Structure . . . 26

3.2 Special Characters and Naming Conventions . . . 29

3.3 Date and Time Format . . . 30

3.4 File Name Format . . . 31

3.5 Modelling Concepts . . . 32

Conventions Concerning Flow Directions and Heights . . . 32

Pressure Reference . . . 33

Flow Controllers . . . 35

Pollutants . . . 36

Effective Flow Rates . . . 38

Histograms . . . 40
Occupants as Source and Occupants as Sensor . . . 42
Default Values . . . 42
Schedules, Time Steps and Simulation Dynamics . . . 42
Notes on Convergence . . . 44

4 Data Sections **47**

4.1 Project Description . . . 48
&-PR-IDENtification (Project Identification) . . . 48
&-PR-UNITS (Project Units) . . . 49
&-PR-SIMUlation (Project Simulation Options) . . . 52
&-PR-OUTPut (Project Output Options) . . . 55
&-PR-CONTrol (Project Control Parameters) . . . 58
&-HISTOgram (Histogram Definition) . . . 61
4.2 Network Description . . . 62
&-CR (Crack) . . . 62
&-NORMCR (Standard condition for crack data) . . . 64
&-FA (Fan) . . . 65
&-DS (Straight Duct) . . . 68
&-F1 (Flow Controller, ideal symmetric) . . . 71
&-F2 (Flow Controller, ideal non-symmetric) . . . 73
&-F3 (Flow Controller, non-ideal symmetric) . . . 75
&-F4 (Flow Controller, non-ideal non-symmetric) . . . 77
&-WI (Windows and Doors) . . . 80
&-TD (Test Data) . . . 87
&-RF (Related Flow Component) . . . 89
&-NET-ZONes (Zones Definition) . . . 92
&-NET-ZL (Zone Layers) . . . 94
&-NET-ZP (Zone Pollutants) . . . 96
&-NET-EXTernal (External Node Data) . . . 98
&-NET-ZT (Zone Thermal Properties) . . . 99
&-NET-LINks (Net Links) . . . 100
4.3 Schedules . . . 104
&-SCH-WINdow (Window Schedules) . . . 104
&-SCH-FANs (Fan Schedules) . . . 105
&-SCH-TEMperature (Temperature Schedules) . . . 106
&-SCH-HUMidity (Humidity Schedules) . . . 107
&-SCH-SINk (Sink Schedules) . . . 108

&-SCH-SOU rce (Source Schedules) . . . 109
&-SCH-OCC upant (Occupant Schedules) . . . 110
&-SCH-MET eo Data (Meteo Data Schedule) . . . 112
&-SCH-POL lutant (Outdoor Pollutant Concentration Schedule) . . . 114
&-SCH-MUL ti schedule data (Multi schedule data) . . . 115
4.4 Building Description . . . 117
&-CP-BUIL ding (Building Reference Height) . . . 117
&-CP-VALU es (Cp Values) . . . 118
4.5 Environment Description . . . 120
&-ENV-BUI lding (Building Related Parameters) . . . 120
&-ENV-WIN d (Wind and Meteo Related Parameters) . . . 121
4.6 Pollutant Data . . . 123
&-POL-DES cription (Pollutant Description) . . . 123
&-POL-FIC tive Sources (Pollutant Fictive Sources) . . . 124
4.7 Occupant Data . . . 125
&-OCCUPAN t (Occupant Description) . . . 125

Appendix **129**

A Examples **130**
A.1 Minimum Input File . . . 130
A.2 Factory . . . 133

B Keywords **139**
B.1 Data Section Keywords . . . 139

C Set-file COMIS.SET **141**
C.1 First part . . . 141
C.2 Second part . . . 141
C.3 Third part . . . 142
C.4 Example . . . 143

List of Figures

1.1 Multizone Structure . . . 9
2.1 COMIS files . . . 20
3.1 Data Groups . . . 26
3.2 Data Structure . . . 27
3.3 Definition of heights . . . 32
3.4 Normal pressure output . . . 34
3.5 Pressure output, outside Reference . . . 34
3.6 Pressure output, outside reference and stack . . . 34
3.7 Flow Controller Characteristic . . . 35
3.8 Keywords for the pollutant transport model . . . 37
3.9 Schedules . . . 43
4.1 Fan characteristic . . . 67
4.2 Fittings . . . 69
4.3 Flow controller ideal symmetric . . . 72
4.4 Flow controller ideal non-symmetric . . . 74
4.5 Flow controller non-ideal symmetric . . . 76
4.6 Flow controller non-ideal non-symmetric . . . 79
4.7 Windows and doors, LVO Type 1 . . . 83
4.8 Windows and doors, LVO Type 1 slanted . . . 83
4.9 Windows and doors, LVO Type 2 and 3 . . . 84
4.10 Windows and doors, LVO Type 2 and 3 . . . 85
4.11 Related Flow Component . . . 90
4.12 Kitchen Hood . . . 91
4.13 Layered Zone . . . 95
4.14 Definition of T-Junction nodes . . . 101
4.15 Definition of link height . . . 102
4.16 Building orientation . . . 120
4.17 Wind Velocity Profile . . . 122
A.1 Scheme Factory Building . . . 134
A.2 Schedule values Temperature, Openings . . . 135
A.3 Schedule values Source, Fan . . . 135
A.4 Pollutant Loads for one day . . . 136
A.5 Pollutant Loads: frequency . . . 136

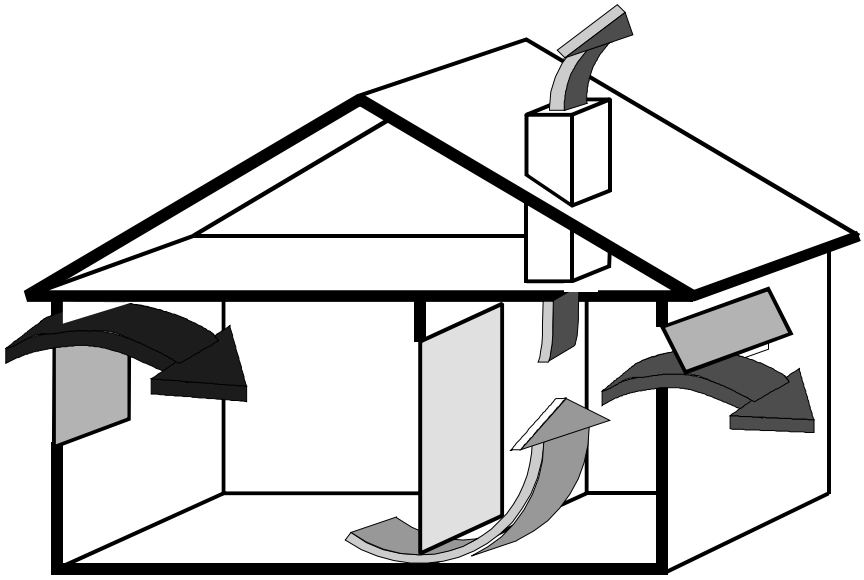


Figure 1.1: Example of a simple multizone structure

1 Introduction

Airflow models can be divided into two main categories, single-zone models and multizone models. Single-zone models assume that the structure can be described by a single, well-mixed zone. The major application for this model type is the single-storey, single family house with no internal partitions (e.g., all internal doors are open). As a large number of buildings, however, have floor plans that would characterize them more accurately as multizone structures, more detailed models, taking internal partitions into account, have been developed.

Even before the advent of physical single-zone models a number of computer models had been developed to calculate the air flow distribution in multizone buildings. In these models the building is described by a set of zones interconnected by flow paths (links). Each zone (node) represents a space with uniform pressure conditions inside or outside the building and the interconnections correspond to impediments to air flow. The network models are usually based on the conservation of mass in each of the zones in the building.

Multizone models are required when there are internal partitions in a building, or in the case of inhomogeneous concentration in the space. Multizone buildings can be either single-room structures (e.g., airplane hangars) single family houses or large building complexes. Figure 1.1 shows an example of a very simple multizone building.

Multizone airflow network models deal with the complexity of flows in a building by recognizing the effects of internal flow restrictions. They require extensive information about flow characteristics and pressure distributions and, in many cases, are too complex to justify their use in predicting flow for simple structures such as single family residences [4].

A literature review undertaken by Feustel and Kendon [17] revealed 26 papers describing 15 different multizone airflow models which had been developed in eight countries. A follow up review [13] carried out in 1992 produced additional information about the status of network models. One of the first to be found was BSRIA's model LEAK by Jackman, which was published in 1970 [21]. Since that time many more models have been developed but many of them have been written as research tools. One of the latest developments in multizone air flow and contaminant transport modeling is the COMIS model [15, 16] which is described here in detail.

1.1 The COMIS Project

Within the Energy Performance of Buildings Group at the Lawrence Berkeley Laboratory LBL (Berkeley, USA), after reviewing the available multizone infiltration models, it was clear that conventional models are not designed to be upgraded to take additional types of flow into account or to improve their usability. Therefore, it was planned to design and develop a model which should contain all the missing features. From the beginning, it was realized that this model development needed a lot of expertise and, equally important, manpower, which would exceed the resources at LBL.

As a consequence, the group made its plan public and asked interested colleagues to join for a twelve month period for the Conjunction of Multizone Infiltration Specialists to lay the foundation of a versatile multizone infiltration model. In 1986, the first COMIS Newsletter was sent to colleagues to inform them about the joint research project planned at LBL. Even though this kind of co-operation is well established in other fields of research, e.g., high energy physics, in the field of building physics it is new to engage in a research project which one individual or country would not be able to do alone. From the beginning the COMIS idea was well received. Owing to the diverse background of the group several national and international research programs were co-ordinated with the COMIS workshop.

This one year COMIS workshop (Conjunction Of Multizone Infiltration Specialists) was held at the Lawrence Berkeley Laboratory from November 1988 until October 1989. A multinational team developed a new multizone infiltration, ventilation and contaminant transport model, named at that time COMVEN. This model not only takes crack flow into account but also covers flow through large openings, single-sided ventilation, cross ventilation and HVAC-systems. The model was designed to consist of a large number of modules which are managed by a steering program. COMIS can also be used as a basis for future extension in order to increase the ability to simulate buildings.

In parallel, a user interface, developed on FORTRAN 77 basis like the COMVEN code, and a result graphical postprocessor programme on LOTUS spreadsheet basis were developed.

1.2 The original COMIS Model (COMIS Version 1.0)

Input

Special emphasis was given to the input/output routines so that the program should be not only "user-tolerant" but "user-friendly". It is being developed so it can be used either as a "stand-alone ventilation model" or as an "ventilation module" of a building simulation program. The input/output procedure is therefore being developed in such a way that either the COMIS input/output modules can be used or only the input/output interface. This makes it possible for the user to connect the program with other software (e.g., the building and systems simulations code TRNSYS).

Flow through Building Components

Crack flow, large openings and mechanical ventilation systems can be modelled by COMIS. Furthermore, additional flows which do not influence the pressure distribution in the network in a major way, i.e., simultaneous two way flow at large openings and wind turbulence effect at single-sided windows etc., were studied. Air flow rates through doorways, windows and other common large openings are significant ways in which air, pollutants and thermal energy are transferred from one zone of a building to another.

However, in a previous review of multizone infiltration models, none of the described codes were able to solve this problem in any way other than to divide the large opening into a series of small ones described by crack flow equations.

COMIS's contribution to this fundamental problem was to describe the physical problem, review the various solutions developed in the literature and compare these solutions using both a numerical

and a physical point of view [10].

The general laws demonstrated by thermal or fluid mechanics approaches are also valid for large exterior openings in steady state conditions. But none of these methods enables us to quantify the effect of an unsteady wind or large scale turbulence.

Experimental results have shown that these effects can be particularly significant in the case of one-sided ventilation. Nevertheless very few correlations have been proposed and most of those that have concern particular configurations. It seems difficult, therefore, to introduce these effects in a general way in our first model. However, we will hope to do so later on as an improvement to COMIS.

The correction of coefficients of power law for crack flow, taking into account the effect of the temperature distribution of air in the crack, is also studied. The temperature of air flowing through a crack depends on the following factors:

- ◇ air flow rate,
- ◇ air temperatures of the zones on both sides of the crack,
- ◇ dimensions and form of the crack.

In most cases the temperature of the air in a crack is quite different from the temperatures of the zones on either side of the crack. Furthermore, air leakage performance measurements are usually performed in a certain temperature condition but used at different temperatures. The temperature variation, however, has a large influence on the air leakage flow due to changes in the air viscosity and air density. Unfortunately, almost all the models dealing with air leakage characteristics ignore this phenomenon.

Data obtained from measurements on crack models show that, for turbulent crack flow, the mathematical description of the friction factor is identical with the one found for conduit flow with smooth walls. Therefore crack flow can be seen as duct flow with a more complicated flow path.

We found from the crack flow equation research that the flow performance is strongly temperature dependent. In order to arrange the results in the usual form we have introduced correction factors which account for the temperature influence. The correction factor depends on the type of leakage. We have developed three different equations for the different correction factors.

We can easily build an air leakage temperature module according to the crack forms. Fortunately, we found that the crack form mainly depends on the structure of the building or on the type of building component and that its size depends on the workmanship. We therefore classify crack forms into three groups: double frame windows, single frame windows and doors, and walls.

HVAC-Systems (heating, ventilating and air conditioning systems) are composed of ducts, duct fittings, junctions, fans, air filters, heating and cooling coils, air-to-air heat exchangers, flow controllers, etc. Several of the program modules concerning ventilating systems have already been developed, allowing us to calculate the coefficients of the flow equation for duct works with fittings, the static pressure losses for T-junctions and the volume flow rate of a fan as well as for a flow controller as a function of the pressure difference. Since the duct systems are described by a network in the air flow model the junction is treated as a pressure node.

There are some data available in the literature for the pressure loss coefficients at the T- junction. To our surprise we found that the values of the pressure loss coefficients were significantly different for different sources. For example, in the case of converging flow, the pressure loss coefficient through the main duct of the T-junction obtained from one source is double the value of the loss coefficient given in another.

The fan performance curve is expressed on the basis of more than three data sets of the volume flow rate, and the pressure difference by the polynomial approximate formula using the least square method.

The pressure loss curve for flow controller is expressed by equations based on data sets of the pressure loss and the volume flow rate. The input data is the driving pressure difference of the flow

controller. The output data is the volume flow rate.

There are other components connected to the HVAC-Systems which cause dynamic pressure loss, e.g., air filters, heating or cooling coils, different types of junctions, etc. Models for these components are not included in the code yet and have to be considered by defining a dynamic pressure loss factor.

Solver

Calculating the infiltration and ventilation flow rates requires the solution of a non-linear system of equations. The main task has been to find an efficient solving method [10], [20].

A building is basically modeled by pressure nodes that are interconnected with air flow links. For one time step, the outside of the building is represented by a fixed boundary condition. The pressures of the internal nodes in the air flow network have to be solved so as to determine the different air flow rates. Solving these infiltration and ventilation flow rates requires the use of a non-linear system of flow equations. The main task was to find an efficient and stable method.

The starting point is the Newton-Raphson method, with derivatives, operating on a node- oriented network which, in most cases, quickly brings about the convergence of the system of equations. The method has been modified to avoid occasional convergence problems when working with power functions. Fortunately, the origin of the convergence problems is well understood. The solving method works on the flow balance equations and not on the flow equations. If one or several of these balance equations have an exponent close to one-half, the Newton-Raphson method will not work well, due to the nature of the procedure, in finding the next approximation. One instance when this happens is when a leakage opening with a flow exponent of one-half is predominant in one zone. In this case the flow balance equation will also have an exponent close to one- half. An under-relaxation will increase the convergence velocity and bring us to the solution. In principle it is a question of finding an appropriate relaxation coefficient.

Wind Pressures

Another important task in the COMIS workshop was to find a method of determining the wind pressure distribution for a building according to measured data from available literature. This allows building designers to work with the COMIS model even if wind tunnel results are not available for the building under consideration. The pressure distribution around a building is usually described by a dimensionless pressure coefficient (Cp), which is the ratio of the surface pressure and the dynamic pressure in the undisturbed flow pattern, measured at a reference height. From experience we know that wall-averaged values of Cp usually do not match the accuracy required for air flow calculation models.

In order to calculate the Cp-distribution for buildings we provide a method based on a parametrical study to determine the Cp-values [10]. The available methods have been checked by comparing calculated results with findings from wind tunnel tests found in the literature. Since the results did not match the data well, a parametrical analysis of wind tunnel test data, aimed at developing a calculation model for Cp-data, was carried out.

The work on Cp-value calculation has been continued later in the AIOLOS project of the European Commission and resulted in the CPCALC+ code [3].

Products of the COMIS workshop

The major products of the COMIS workshop are:

- ◇ COMIS simulation code COMVEN
- ◇ COMIS User interface COMIN
- ◇ COMIS Output processor COMOUT
- ◇ COMIS User's Guide

- ◇ COMIS Fundamentals [14]
- ◇ Special Issue of the Journal Energy and Buildings [10]

1.3 Follow-Up of COMIS: IEA Annex 23

Work on the COMIS program did not finish by October 1989. A computer code was available but such a program is ever perfectible. The validation procedure itself is a huge work and was not completed during the COMIS year. Moreover, new knowledge (e.g. from IEA Annex 20) was available after the COMIS workshop finished. This knowledge had to be integrated into the COMIS code.

A roundtable discussion between the COMIS participants and the COMIS review panel about future perspectives revealed a strong feeling that COMIS (or its successor) ought to operate as an international institution with participants committing themselves to a definite work load.

The COMIS group suggested to start a working group under the hospice of IEA's Energy Conservation in Buildings and Community Systems program. The *Multizone Air Flow Modeling* working group was officially adopted in June 1990 as Annex 23. The Annex was supported between 1990 and 1996 by nine participating nations: Belgium, Canada, France, Greece, Italy, Japan, Switzerland, The Netherlands, and USA. The objective of this Annex was to study physical phenomena causing air flow and pollutant transport (e.g., moisture) in multizone buildings and to develop modules to be integrated in a multizone air flow modelling system. Special emphasis was given to the comparison between results from the model and from in-situ measurements.

To reach these objectives the IEA Annex was structured in three subtasks:

- ◇ System development (subtask 1)
- ◇ Data acquisition (subtask 2)
- ◇ System evaluation (subtask 3)

Results of these subtasks are addressed to researchers and consultants and contribute to the promotion of an energy efficient design. The Annex participants undertook a task sharing project involving model development, data acquisition and analytical studies.

A close cooperation was envisaged, mainly with regard to state-of-the art reviews, data collection, coordination of work, e.g. defining cases for evaluation purposes with other pertinent projects. The Air Infiltration and Ventilation Centre acted as a vehicle for disseminating the results of this particular Annex. A data base for evaluation purposes was prepared by AIVC and the Centre has collected wind pressure data and leakage data. Algorithms developed by Annex 20 were incorporated into the modelling system. Data obtained for evaluation purposes by this Annex was used for subtask 3. The overlapping in time with Annex 20 guaranteed a sufficient transfer of knowledge.

Products of IEA Annex 23

The major products of IEA Annex 23 are:

- ◇ COMIS 3.0 with graphical IISiBat/COMIS user interface
- ◇ COMERL user interface
- ◇ User Guides for COMIS, IISiBat/COMIS and COMERL
- ◇ Annex 23 Report 'Evaluation of COMIS' [18]
- ◇ Annex 23 Report 'Evaluation of COMIS' [19]
- ◇ Special Issue of the Journal Energy and Buildings [11]
- ◇ COMIS Programmer's Guide [8]
- ◇ COMIS Sample Runs [25]

The COMIS group is also present in the WWW under http://epb1.lbl.gov/comis/. From this site, there is also the possibility to download the Annex 23 version of COMIS (COMIS 3.0).

The IIsibat/COMIS 3.0 version can be downloaded from the web under http://evl.cstb.fr/francais/projets/IISiBat/COMIS/comis1.htm, together with a handbook for the user interface and with a tutorial.

1.4 COMIS version 3.1

In 1999, EMPA has officially taken over the maintenance of the COMIS source code from LBNL. LBNL will continue to contribute to the further development of COMIS, but LBNL is not directly involved in the maintenance task anymore. CSTB remains responsible for the IIsibat user interface.

Towards the end of Annex 23, additional contributions to the COMIS code were made in the frame of IEA Annex 27 "Evaluation and Demonstration of Domestic Ventilation Systems". Many valuable features were included in the code. However, not all parts were working properly, and the consistency between code and COMIS user guide had diminished noticeably. Together with a list of bugs and other inadequacies in the code, there was a strong need to further improve the COMIS product.

Therefore, EMPA has taken the initiative to update COMIS. With COMIS 3.1 many bugs are fixed, the input formats are consistent and somewhat streamlined where necessary, and the User Guide reflects what the program requires and does.

The update would not have been as profound and extensive without the valuable contributions and help by

- Peter G. Schild (NBI, Norway)
- Brian Smith and Dave Lorenzetti (LBNL, USA)
- Hans C. Phaff (TNO, The Netherlands)
- Werner Keilholz (CSTB, France)

Changes from COMIS version 3.0

New features, improved features and features which were implemented in the code, but were not yet documented in the User Guide:

Simulation Options: page 52
`&-PR-SIMU`
New option to force pollutant transport output with predefined time step:
`POLSTEP step time step2`

Output Options: page 55
`&-PR-OUTP`
New options:
`On` number of Occupant Type n per zone
`Yn` activitY level factor of occupant Type n per zone
`Fn` effective Flow with fictive source n per zone

Control Parameters: page 58
`&-PR-CONTrol`
New solver: solver no. 6.
Data block 2 changed: Only one flag for initialization at the first and one for the subsequent timesteps and the number of iterations are input.

Histograms: page 61
`&-HISTO`
Only parameters for the histogram definition. Definition of fictive sources moved to new keyword `&-POL-FIC`. Occupants can be used as weighting factors in histograms.

Pollutant Description: page 123
Continuation line:
Definition of fictive sources moved to new keyword `&-POL-FIC`.

Fictive Source Definition: page 124
`&-POL-FICtive sources`
New keyword for the definition of fictive sources, used for calculation of effective flows.

Zone Layers: page 94
`&-NET-ZL` has only five input parameters

Schedules: pages 110, 109
`&-SCH-OCCupant`, `&-SCH-SOUrce`
Occupant source schedules solely to be defined under `&-SCH-OCC` keyword;
Number of occupants implemented in `&-SCH-OCC`.

Wind and Meteo: page 121
`&-ENV-WINd`
Data block 2 has only two input parameters: wind direction angle and wind velocity profile exponent at building location.

Occupants: page 125
`&-OCCUPANt`
New data block 2 with data for the source strenth of different pollutants if occupant is used as source. Data in data block 1 starts with *No.

New Histogram output file: page 55
Histograms are not longer written to COF-file, but to a separate file (extension CHO).

Naming convention: page 20
<name>-xy.cso <name>-xy.cho are used for output files instead of xy<name>.cso xy<name>.cho

There were a number of features implemented in the code which still need further development of algorithms and code and thus are not described in the present user guide. Another number of features which are obsolete but which were still listed in the user guide, were of course also removed in the user guide text. The removed or not operational features are:

- ◇ Duct Fitting
- ◇ Passive Stack
- ◇ Transition
- ◇ HVAC Component
- ◇ Cp Calculation

The following keywords are not needed any more

- ◇ `&-CIF`
- ◇ `&-NET-AIR`

COMIS 3.1 Documents

Valid COMIS documents after successful installation under the IISiBat-directory '\icomis' in the COMIS-directory '\icomis\comis\docu'.

UserG31.pdf: User Guide for COMIS v3.1 simulation engine; this document.

COM-TRN.pdf: User Guide for coupling COMIS and TRNSYS.

OutpOpt.pdf: Definition of COMIS output quantities.

ProgGuid.txt: Guidelines how to document COMIS program modifications.

Headers.txt: Sample headers for all keywords; identical with the ones used throughout this user guide.

How to get COMIS 3.1

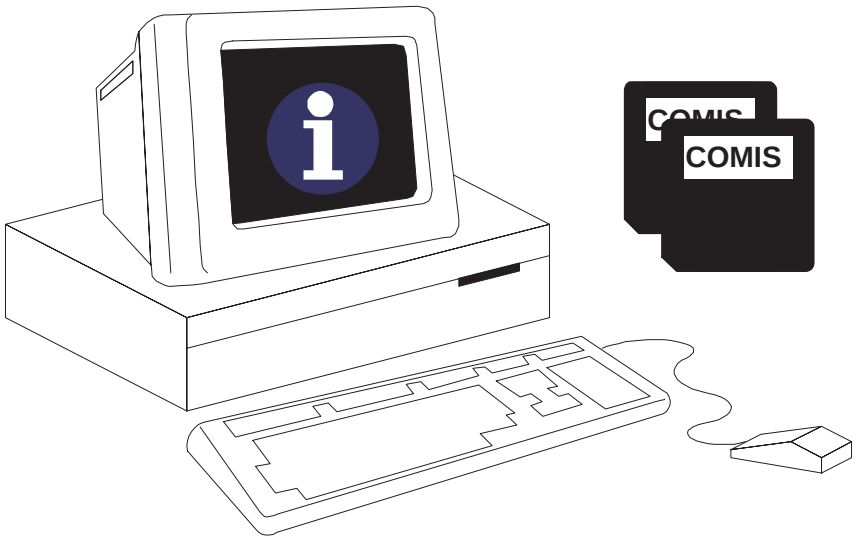
The new COMIS version is available from CSTB, Sophia Antipolis, France. For more information, see

`http://evl.cstb.fr`

and

`http://www.empa.ch/englisch/erg/comis/comis.htm`

2 Getting Started



2.1 Nomenclature in COMIS User Guide

Typesetting

`text`	text from file or command line
`<...>`	input to be supplied by the user
`[...]`	optional input
`|`	alternative options
	"Return" key

Significant characters in keywords

UPPERCASE	required
lowercase	optional

text in margin	hints

2.2 Distribution

The COMIS with IISiBat distribution on CD contains the all files necessary to run COMIS in stand-alone mode (without user interface). The COMIS simulation engine is a Win32 application. It can be executed on MS-Windows 9x, Me, NT, 2000.

To use COMIS under a different operating system the source code of the COMIS simulation engine has to be recompiled for that system.

If COMIS is used in stand-alone mode, the input file either already exists or will have to be developed with a text editor. The procedures described here are valid for the use of the program without an interface.

Several user-interfaces have been developed for COMIS (e.g., IISiBat and a Excel-GUI). A short description can be found in section 2.5, pages 22ff. The information has been taken from online help and short guides coming with the interfaces. For program usage with an interface, please refer to the user's guide provided by the interface developer.

2.3 Installation

Installation on a PC

For installation on a PC, follow the instructions coming with the COMIS with IISiBat package distribution CD.

After a successful installation, the COMIS files can be found under the IISiBat-directory '\icomis' in the COMIS-directory '\icomis\comis' and its subdirectories:

```
\icomis
  |________Appli
  |________Comis                    COMIS directory
  |         |________Binary         COMIS v3.1 Executable
  |         |________Docu           User Guide, etc
  |         |________Example        Example input files, SET-file
  |         |________Source         Fortran source code
  |________...
  |...
```

No other files than those in the COMIS-directory are necessary to run COMIS in stand-alone mode (without user interface).

Compilation of COMIS

The COMIS executable on the COMIS with IISiBat package distribution CD was produced with COMPAQ Visual FORTRAN 6.1. It can be executed on MS-Windows 9x, Me, NT, 2000. To port COMIS to other platforms it has to be recompiled with a corresponding compiler.

The COMIS source is subdivided into almost 60 files named 'COMV-*.FOR' and four include files named 'COMV-*.INC'. After the installation of COMIS with IISiBat they can be found in the directory '\icomis\comis\source'. Compile and link them all together into one executable file. The subroutine '`RdCmdLn`' located in the source file 'COMV-WIN.FOR' reads the optional command line parameters of COMIS. As this function depends on the operating system and the fortran compiler, there are some code lines in '`RdCmdLn`' which have to be adjusted to the desired compiler and operating system.

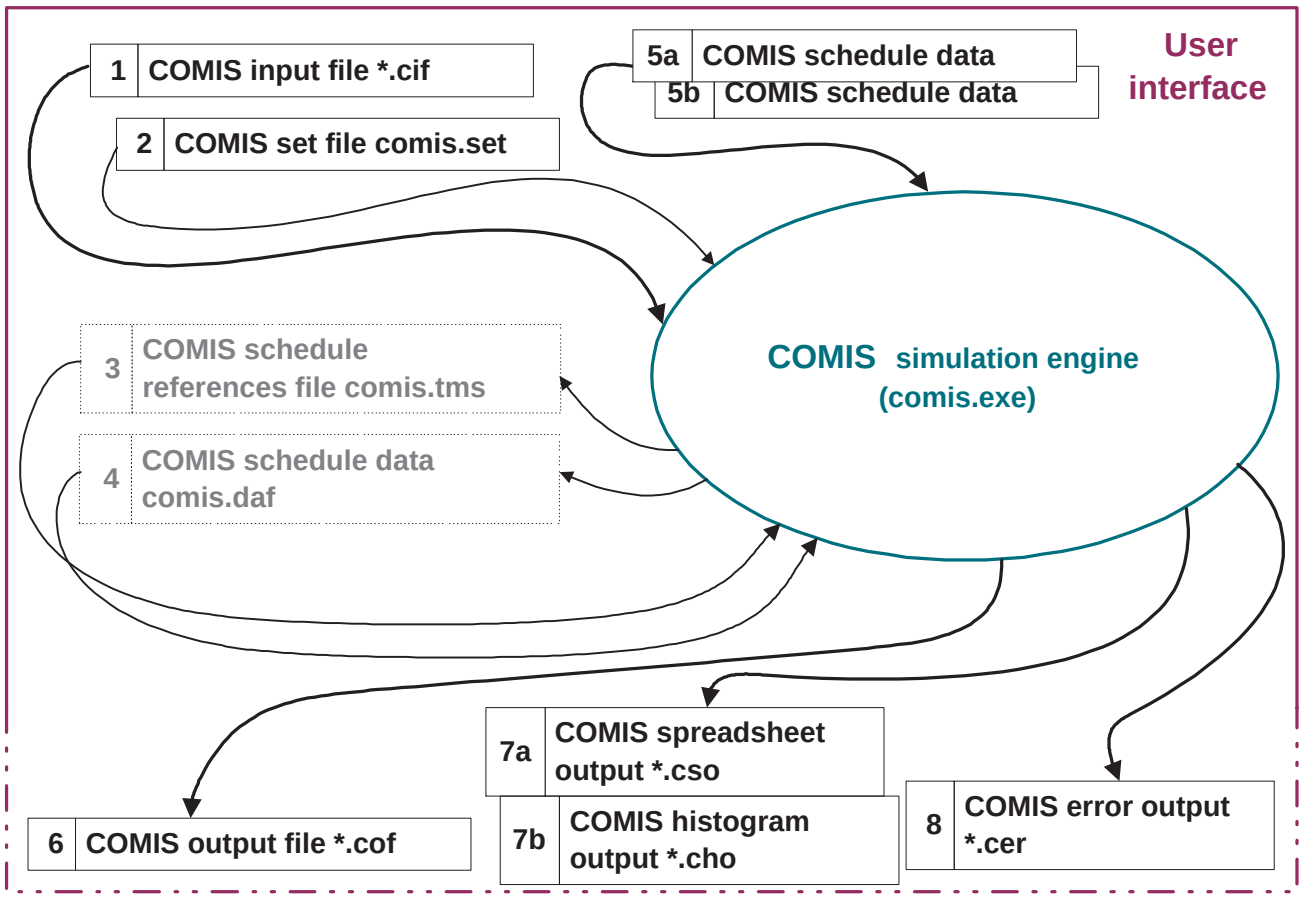


Figure 2.1: COMIS files. Arrows denote information flow.

2.4 COMIS File Structure (Stand-alone installation)

COMIS uses the files shown in figure 2.1 for input, output, and temporary storage.

File name and extension[1]	I/O[2]	Defined[3]	Description
<name1.cif>	in	cls	Input file with simulation model description
comis.set	in	—	Set-file for expert options[4]
<name5a.cmf>	in	cif	Meteorology schedule data
<name5b.csm>	in	cif	Multi schedule data
<name5c.csf>	in	cif	Other schedule and data files
comis.daf, comis.tms	tmp	—	Internal COMIS temporary time management files
<name6.cof>	out	cls	Standard output file
<name7>-xy.cso	out	cls	Spreadsheet output files
<name7>-xy.cho	out	cls	Histogram output files
comis.cer	out	—	Listing of error messages

The only files that inevitably must exist to run COMIS are the executable and an input file.

[1] Extensions included in brackets may be choosen by the user. However, it is recommended to use the extensions given here.

[2] cls: file name defined on command line or in COMIS.SET; cif: file name defined in input file

[3] in: files read by COMIS simulation engine; tmp: temporary files, out: output files

[4] A description of COMIS.SET can be found in Appendix C on page 141

When run from the command line, input and output file names (including paths) can either be explicitly given in the program call or can be set in the set-file COMIS.SET. The set-file is searched for in the current directory.

However, because unit settings in the set-file are hard to control, the use of the set-file is strongly discouraged (for details on using / not using a set-file, please confer appendix C).

If no COMIS.SET file is used, the names for the input and output file(s) have to be given on the command line. The syntax of the program call is

`C:\>[exe-path]COMIS31 [options]` ↵

`[exe-path]` is the path to the directory of the executable. In a standard COMIS with IISiBat installation, `[exe-path]` had to be replaced by `icomis\comis\binary\`.

Valid options under Unix or Windows are:

`-a`	`<name>`	Input file is `<name>.cif`, and all output files get the common (project) name `<name>` (path may be included); overrides definition in COMIS.SET file
`-i`	`<file.ext>`	Input filename (including path, usually with extension .CIF)
`-o`	`<file.ext>`	Text output file (including path, usually with extension .COF)
`-t`	`<name>`	Spreadsheet output project name (no path, no file extension)

The file names `<file.ext>` as well as the file names composed with `<name>` must result in valid file names of the respective operating system (i.e. long file names are allowed under Windows 9x and NT).

The '-a' option defines a common (project) name for input and output files. It is useful either to override settings in the set file or to shortcut command line input. Entering

example `C:\>icomis\comis\binary\COMIS31 -a  c:\icomis\comis\files\proj1` ↵

would read input from `proj1.cif` (which must exist in advance), and send output to `proj1.cof` and spreadsheet output to `proj1-xy.cso` (with `xy` a valid spreadsheet output option defined in `proj1.cif`, see section 4.1). The files will be found in `c:\icomis\comis\files\`

If no SET-file is present, at least the input file and output option `-o` have to be given. If COMIS is called without any options and if no specifications for input and output files are made in COMIS.SET, an error is reported.

2.5 COMIS User Interfaces

COMERL SHELL User Interface

This section is obsolete. COMERL will not be updated for COMIS version 3.1.

IISiBat User Interface

This program uses a GUI to create COMIS input files for COMIS 3.1 and allows to execute simulations and to analyse the results. Documentation comes along with the COMIS 3.1 package distribution.

- Project Creation

  The Assembly Panel is the window in IISIBat where the user will create, modify, and run assemblies of models (projects). The Assembly Panel can be accessed by clicking on the COMIS icon in the main IISIBat window or by clicking on an existing project in a project library. Every Assembly Panel has a toolbox of assembly tools.

  - Model instanciation

    To define a project, you first have to instanciate COMIS models (components) in the assembly panel (drag & drop from library).

    Next, links representing air flow between the models have to be defined. This is done with the link tool in the assembly panel.

    To edit the parameters of a model, click on the upper part of the model's icon.

  - Attachments

    Certain models, such as pollutants or temperature schedules, can be attached to other models. To do so, one has to click on the lower part of the model's icon. This opens a window allowing to edit the attachments of the model.

  - To get started

    Some sample projects are available in the standard libraries. They show how COMIS components are typically linked.

- Simulation

  To start a simulation, one clicks on the run tool in the assembly window. This creates the COMIS Input File (CIF). It then starts COMIS using this file.

  To visualize the COMIS Output File and other files created by COMIS, you can use the built-in editor. By default, COMIS runs with IISiBat produce a file called iisibat.out.

COMISExcel User Interface

A spreadsheet user interface has been developed by Peter Schild, Norwegian Building Research Institute. Documentation is included in the downloadable ZIP-file (see below).

A file COMIS.XLS is supplied that serves as template spreadsheet user interface for Microsoft Excel 97. Important features according to Peter Schild include:

- Spreadsheet for data input to COMIS 3.x kernel (generates a standard *.CIF ASCII input file)
- Hopefully simpler, more flexible and robust than other COMIS user interfaces (Input variables can now be calculated in-situ and/or linked to other cells in the spreadsheet)
- Bubble text makes the spreadsheet self-explanatory, with detailed description for most variables
- Can automatically run COMIS solver from within Excel. It is possible to automatically run multiple cases, either for parametric analysis or Monte Carlo calculation.

- ◇ The correct units are automatically shown for each input variable (i.e. hyperlinked to datablock where input/output units are designated)
- ◇ A debugging macro generates a summary table of links (see screen shot at bottom of page)
- ◇ Neat printout of input data file

The latest information on COMISExcel can be found on

`http://www.byggforsk.no/hybvent/COMIS.htm`

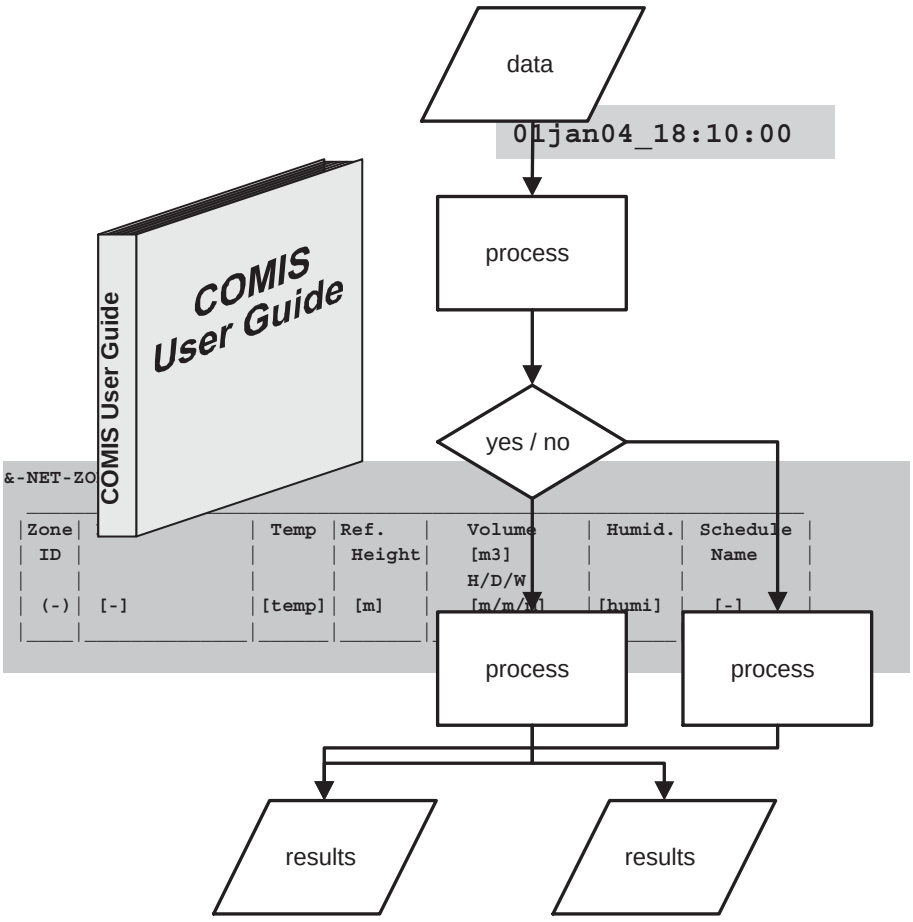
The free download, see next section, "COMIS Tools".

2.6 COMIS Tools

Tools like COMISExcel, a weather data converting utility and others are downloadable from the COMIS eGroups web page,

`http://www.egroups.com/files/comis/`

3 Basic Concepts



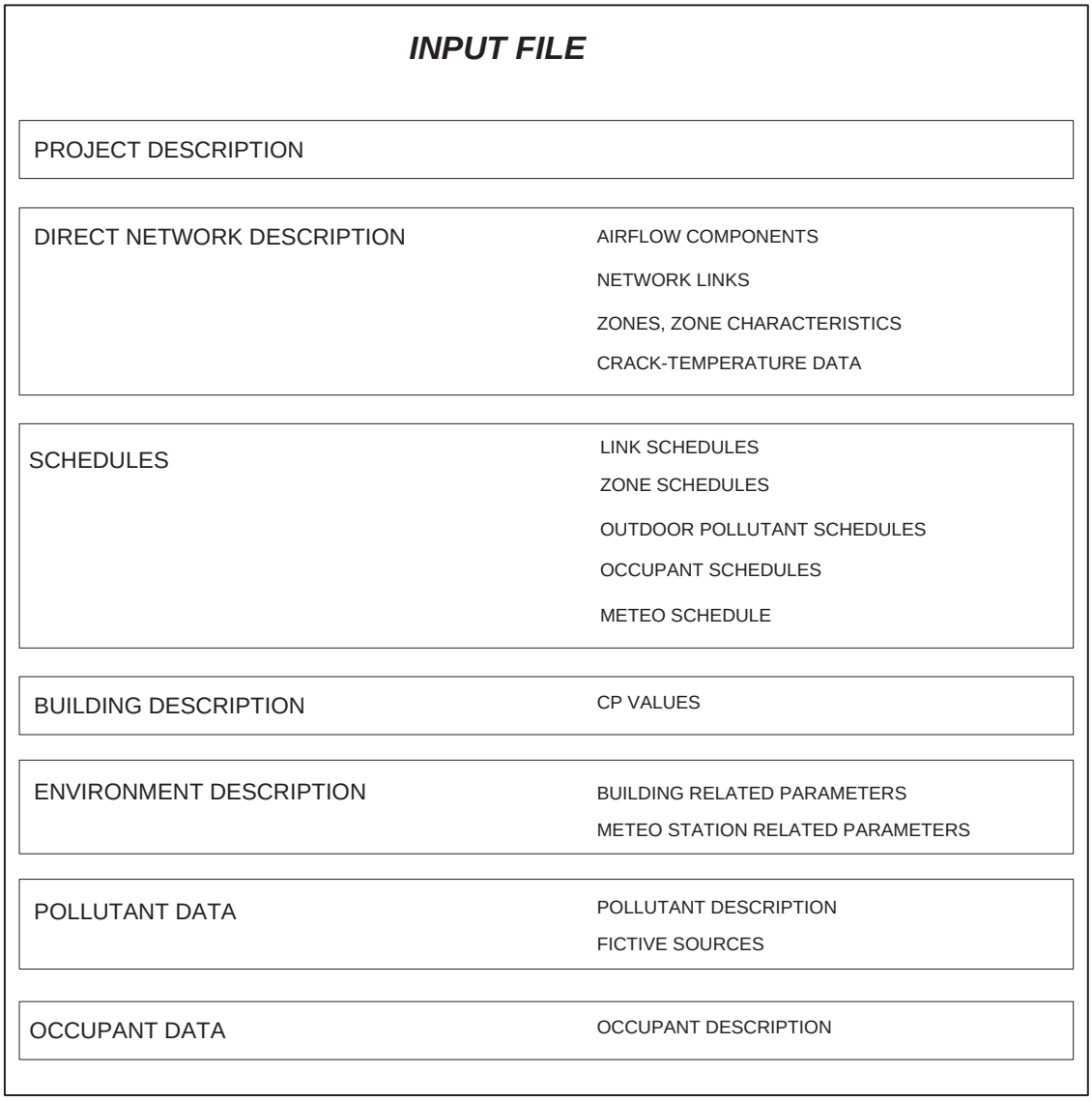


Figure 3.1: Data Groups

3.1 Input Data Structure

Input data in the CIF-file is arranged according to the following classification:

Data group: Data groups are Project, Network, Schedules, Cp-Values, Environment, Pollutant and Occupant (see figure 3.1). Groups are used to improve the user's overview over the necessary data, but the grouping of data need not be reflected in the structure of the input file. The data sections are presented by group in this guide.

Data section: The description of the simulation model in the input file is divided into data sections. The beginning of each section is characterized by a specific keyword. Keywords are characterized by having an `&` in the first column (example: `&-PR-IDENTification`, `&-CR`). The structure of data sections is shown for the example in figure 3.2.

To run a simulation, at minimum one air flow component (from data sections `&-CR` on page 62 to `&-TD` on page 87), one zone (data section `&-NET-ZON` on page 92) and one link (data section `&-NET-LIN` on page 100) must be defined. An example is given in Appendix A.1 on page 130.

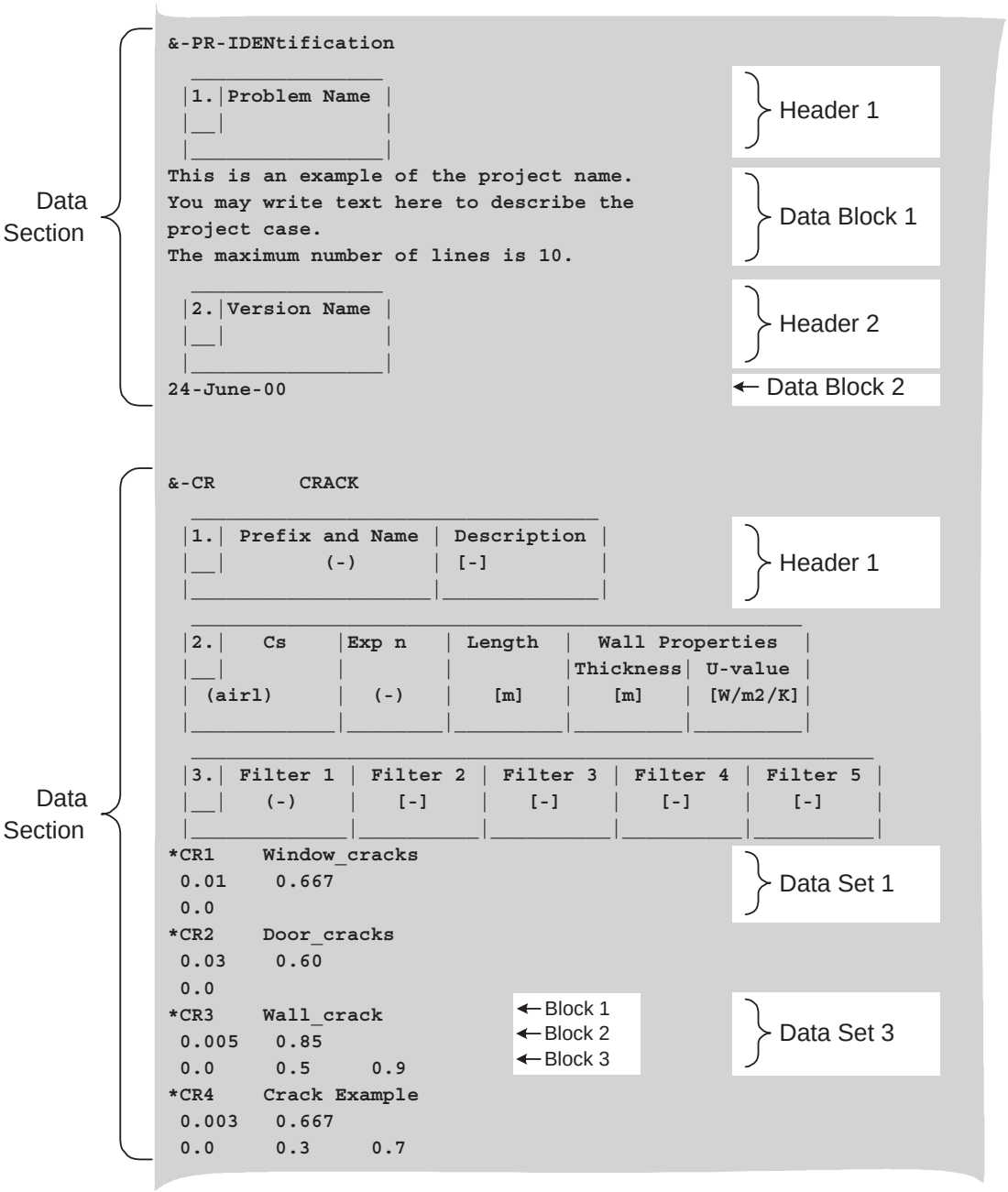


Figure 3.2: Data Structure

Data sections not needed for the actual simulation or consisting of optional parameters only may be entirely omitted. If necessary, internal default values are used to complete the input (e.g. control parameters, ambient conditions, type of output).

Data block: A data section consists of one or more data blocks. A data block has an optional header and can have several data lines.

NOTE Data blocks in a data section or data set must be given in the sequence described by the order of the corresponding headers in section 4 in this guide.

Header of data block: For each data block, a header may be given in the input file. The header

usually contains information on the data required in the data block: parameter names, units and possibly some further descriptions. This allows for the content of the input file to be self-explanatory. Headers are surrounded by vertical and horizontal bars.

A header starts with two blanks followed by a row of underscores '__...'. Headers are, preferably, placed close under the *keyword*.

Headers are provided by the IISiBat user interface coming with the official distribution of COMIS. The COMISExcel uses headers, too. See section "COMIS User Interfaces" on page 22 for more information. Headers (but not the keywords!) may be omitted by the user while establishing an input file using the screen editor. The headers shown in the keyword description later in this guide are stored in the COMIS simulation engine documents directory in a file called 'HEADERS.TXT'.

The units reported in these headers are identified either

- ◇ by a 4-character name of a unit set in section `&-PR-UNITS` or in the set-file, if the unit may be set by the user;
- ◇ by a conventional unit name, if the unit cannot be changed;
- ◇ by a dash, if the quantity has no unit;
- ◇ or by a questionmark, if quantities with eventually different units are possible (example: histogram).

Care has to be taken that values in a particular input file correspond with the units setting in the COMIS.SET file and in the input file (see section `PR-UNITS` on page 49 and appendix C)!

The type of brackets used for the units in the headers indicates whether the specific parameter is required or optional:

`(..)` Required parameter or set of parameters
`[..]` Optional parameter or set of parameters

Data lines: The separator for data items is one or more blanks, a comma or both. A data-line is not allowed to start with a comma.

Data-items in one data line must be given in the sequence described in the header. But they do not have to be arranged in columns. I.e., they do not have to be placed exactly under the respective header. It is, however, advantageous to use the header as a formatting aid.

Only optional parameters following the last mandatory parameter may be omitted entirely. Otherwise some value must be put at the place of an optional parameter preceding mandatory parameters or optional parameters that shall have an other than the default value.

Any abbreviation of the word 'default' (without a period '.') may be used in a place where the user wants to use a default value, or to skip over an optional parameter. Examples are 'd', 'def'. Uppercase and lowercase are not significant.

Data set: In many data sections, more than one element can be defined. Then, data blocks belonging to one element are grouped together. Data sets are a set of data blocks defining a (physical or logical) element of the simulation model. The first item of the first block in a data set is a specific `*<name>`, `<name>`, `*<no>` or `<no>` with which the data set can be identified and referenced. In the example shown in figure 3.2 several cracks are defined.

3.2 Special Characters and Naming Conventions

The special characters used are:

'&'	as first non-blank character in a line, a keyword follows in this line
'#' or ';' or '|' or '_'	as first non-blank character in a line, line is regarded as a comment
'#' or ';'	after some input in a line, rest of line is regarded as a comment
'|', '_'	the headers are surrounded by bars '|' (ASCII char 124) and underscores '_', (ASCII char 95)
'_'	delimiter in date_time string
':'	(optional) delimiter in time string
'/'	delimiter for parameters where a parameter list is allowed in the place of a single value
'-'	parameter range indicator where a parameter list is allowed in the place of a single value
'*'	as first non-blank character in a line, preceding a name or number without blank space in between, indicates that all lines until next line beginning with '*' or until next keyword belong to the definition of the actual data set
'*'	in the multi schedule or meteo schedule file: as first non-blank character in the first line '*' indicates that a point in time and a time period follows
'␣', ','	data delimiter in data lines
'd[e[f[a[u[l[t]]]]]]'	if placed in a data line where a value is expected, an internal default value is taken as value for the respective parameter

A '#' inserted at the beginning of a data line converts this line to a comment. This makes it easy to change a line by copying it into a new line and yet keeping the old line with '#' before the first data element.

Network elements that are referenced by other components in COMIS have an identifier (a name, or, in certain cases, a number). Generally, names may consist of any character sequence. Not allowed are the characters '/' and '-' because they are used as delimiters in lists of names. If an element with a name as identifier is used where a parameter list is allowed, the use of the above mentioned characters would lead to a misinterpretation of the name.

3.3 Date and Time Format

Date and time input is required in several data sections in COMIS. Date and time inputs in COMIS are interpreted as a point in time where a certain event occurs.

Date and time are entered as date_time string. Date_time strings consisting of date or time only are allowed.

Valid `date_` expressions have one of the following formats:

```
MDD_       MMDD_      YYMMDD_    YYYYMMDD_
monDD_     monDD_     YYmonDD_   YYYYmonDD_
day_
```

in which
- `M` digit of the number of the month
- `D` digit of the calendar day number
- `Y` digit of the year number
- `mon` name of the month or valid abbreviation (minimum 3 characters);
  e.g. jan; JAN; januar; january; feb; februa; mar; apr; may; jun; jul; aug; sep; oct; nov; dec
- `day` name of the day of the week or valid abbreviation (minimum 3 characters);
  e.g. mon; tue; WED; thu; fri; sat; sun; WDY; wnd.

Names of month and day are case insensitive, so `JAN`, or `Jan`, for January would work, too.

`WDY_` and `WND_` correspond to weekday (monday til friday) and weekend (saturday and sunday), respectively. For weekday and weekend, no other abbreviations are possible.

In order to distinguish between the date and the time, the date is followed by an underscore character '_'. The delimiter ':' can be used to separate hours, minutes and seconds in the time expression.

Valid formats for the time input are:

```
h     hh     hmm     hhmm      hmmss      hhmmss
h:    hh:    h:mm    hh:mm:    h:mm:ss    hh:mm:ss
```

in which
- `h` digit of the hour
- `m` digit of the minute
- `s` digit of the second

The date_time expression must not contain any blanks.

Following are some examples of how the data for date and time might be entered into the program, and the interpretation of the date_time expression:

`89aug04_`	or	`890804_`			2089 august 04 at 00:00
`1989aug04_`	or	`19890804_`			1989 august 04 at 00:00
`aug04_`	or	`0804_`	or	`804_`	august 04 any year at 00:00
`aug04_18:23`	or	`0804_18:23`	or	`804_18:23`	august 04 any year at 18:23
`SUN`					every sunday at 00:00
`FRI_18:00`	or	`FRI_18`			every friday at 18:00
`18:23:10`	or	`182310`			every day at 18 hours 23 minutes 10 seconds
`18:23:00`	or	`18:23`	or	`1823`	every day at 18 hours 23 minutes 0 seconds
`18:00`	or	`18:`	or	`18`	every day at 18 hours 0 minutes 0 seconds

3.4 File Name Format

In many data sections the user may define a data file instead of giving explicit input data in the CIF-file. The format of this file allocation is:

```
F: <filename>
```

or

```
F: <schedule name> <filename>
```

The second form has to be used to connect a file which contains data for referenced schedules. `<filename>` can be any file designation in compliance to the specific computer operating system requirements.

Example of a Cp-value data file assignment:

for DOS- and Windows System:	`F:C:\comis\exampl\comin\cp1.dat`
for UNIX-System:	`F:~comis/@exampl/comin_cp.cp1.dat`
for VMS -System:	`F:[username.comis.exampl]comin_cp_cp1.dat`

The content of these files must be formatted exactly according to the requirements set up in the following sections for the specific data.

3.5 Modelling Concepts

COMIS is based on a node model of a building. It is assumed that each zone is homogeneous, and can be represented by a node with single values for temperature, pressure, and possibly for each contaminant concentration. The nodes are linked by non linear conductances, modelling the air paths (cracks, openings, ducts, etc.). Pressure coefficients, relating the wind pressure at the building to the wind velocity, can be attributed to external nodes. Not only wind effect, but also stack effects resulting from temperature and air composition differences, as well as fans, are taken into account. Using air mass conservation, a system of non linear equations is built and solved. The physical models used are described in [14].

In this section, explanations referring to more than one keyword and important features and underlying concepts of the models are given. In addition, the equations for fictive sources and effective flow rates, and the calculation of the frequency distribution for COMIS histograms are documented here, because they have not been documented elsewhere.

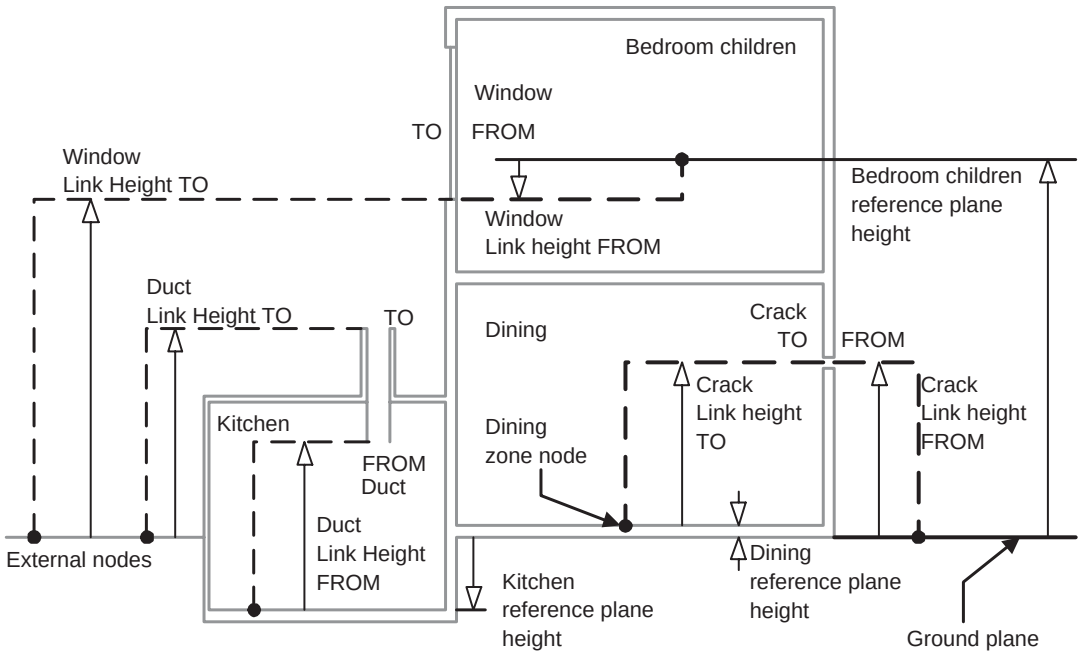


Figure 3.3: Definition of heights. Example zones from `&-NET-ZON`, see page 92.

Conventions Concerning Flow Directions and Heights

A link is defined between two zones, see `&-NET-LIN` on page 100. Flows have a positive sign in direction from zone `From` to zone `To`. For most airflow components it is arbitrary which zone is chosen to be `From` and which is chosen to be `To`. The exceptions are

- ◇ T-junctions, see `&-NET-LIN` on page 100
- ◇ Active elements, like fan, and – eventually – test data components
- ◇ Flow controllers.

Care has to be taken that one gets the desired flow directions for these components!

For passive elements, which constitute a simple resistance to air flow, the pressure decreases in flow direction. For such elements, the pressure difference in COMIS is defined as

$$\Delta p = p_{\mathrm{From}} - p_{\mathrm{To}}$$

Active elements like fans push the flow against the pressure difference. To have positive pressure values, COMIS uses

$$\Delta p = p_{\mathrm{To}} - p_{\mathrm{From}}$$

as pressure difference for fans.

The Test Data component is supposed to be used for both passive and active elements. Its pressure difference however is that used for passive elements. I.e., if fans are to be modelled with a Test Data component, negative pressure differences correspond to positive air flows.

To account for the thermal stack pressure in the right way, the vertical position of zones and links must be known, see figure 3.3. In COMIS, a reference plane has to be defined for every zone. This plane is characterized by its height relative to the ground plane. The reference plane for a zone can be chosen as is convenient for the project. The only exception are layered zones, where the reference plane has to coincide with the floor plane. The reference plane for the surroundings of the building is the ground plane.

NOTE Internal (i.e. zone) nodes are located in the zone reference plane. External nodes are located in the ground plane.

Links are also situated in a certain height. Their position is measured relative to the zone reference plane. If the reference planes of the `From` and the `To` zone of a link are different, the `Height From` and `Height To` of this link will differ, too.

The height definitions are illustrated in figure 3.3

Pressure Reference

COMIS normally uses pressures with reference to the outside pressure (without wind) at the reference level of the building. This means that the pressure decreases about 12 Pascal (Pa) per meter height, as in reality. This is due to the gravity field. Each cubic meter of air weighs about 12 N, which results in the 12 Pa/m pressure change.

Consider two rooms, one on top of the other (see figure 3.4).

If there is no flow through the cracks of the ceiling between the rooms, then the pressure in the upper room is about 36 Pa lower than in the lower room. Here it is about 33 Pa due to the altitude of the building of 588 m. Both room pressures (-10.7 and -43.9 Pa) are defined here at the floor level and the ground plane is one meter below the floor of zone 1. Looking at the zone pressures, in figure 3.4, one might first think that there must be a flow from zone 1 to zone 2. However this flow is zero. At a height of 3 m in zone 1 the pressure equals the pressure in zone 2 at 0 m.

One could say: as there is no flow between the two rooms their pressures are equal, therefore they must be the same in the COMIS output. A way to do this is simple: use as reference pressure the outside pressure (without wind) at the altitude of the zone reference height (see figure 3.5).

If the characters '-OSR'(take "Outside Stack as Reference") are attached to the output units for pressure (e.g. 'Pa-OSR') under the `&-PR-UNITS` keyword (see page 49) then the corrected zone pressures are printed.

The pressures now almost resemble the pressure hierarchy. However if the rooms have a temperature that differs from outside, the thermal stack pressure (with respect to outside) can still result in a small pressure difference between the rooms that seems not to agree with the flow direction at first glance (see figure 3.6).

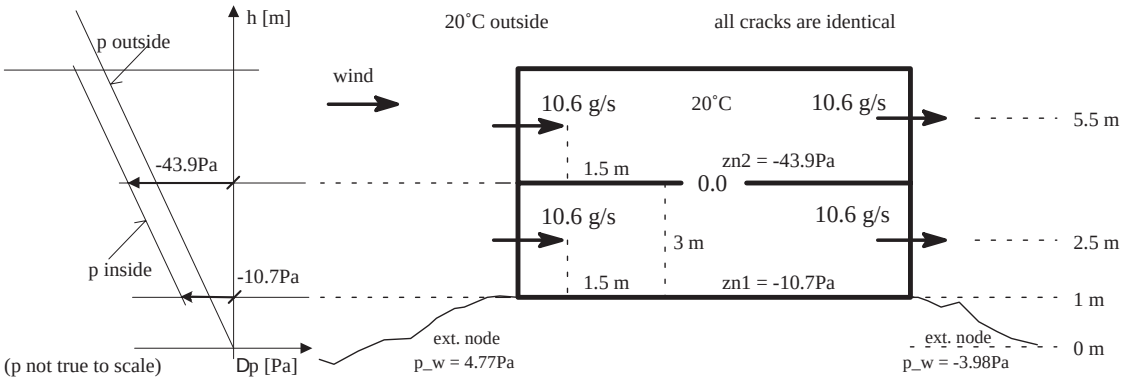


Figure 3.4: Two zones with normal pressure output.

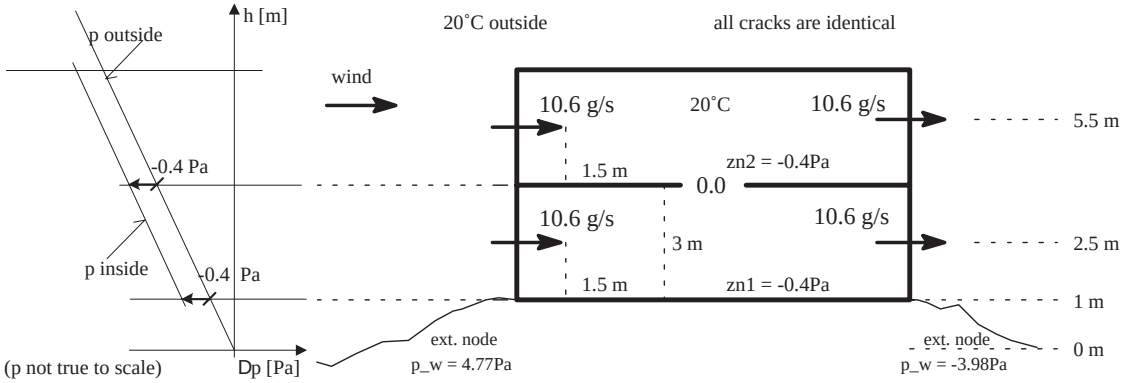


Figure 3.5: Same as figure 3.4, but with outside pressures at zone reference height as reference.

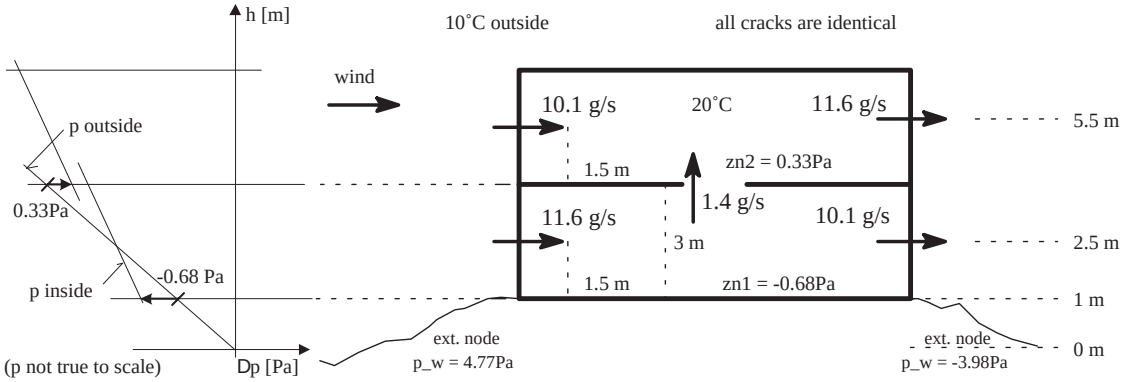


Figure 3.6: As figure 3.5, but with a 10K stack pressure difference.

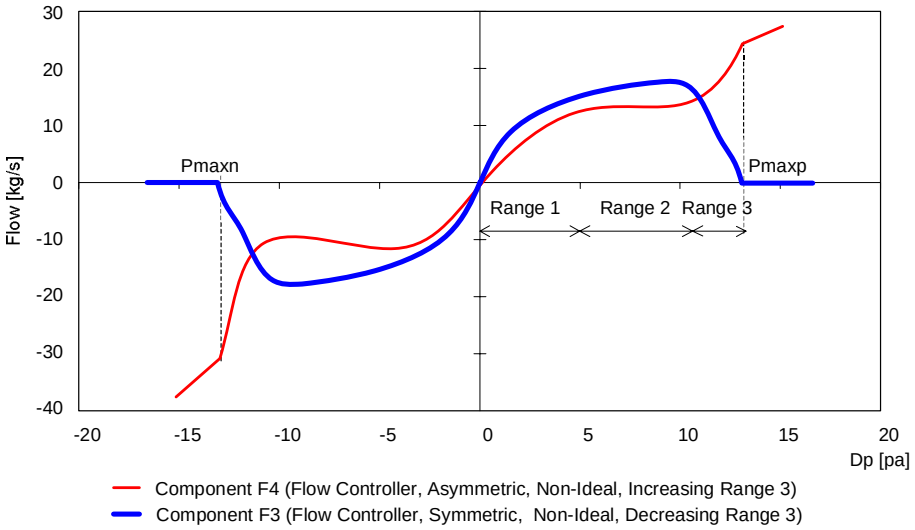


Figure 3.7: Characteristic of flow controllers

Flow Controllers

In COMIS, four types of flow controllers are available which represent most of the available dampers or regulators with input signal from the pressure drop or (duct) flow. Controllers with input signal from temperature must be simulated with the schedules.

The basic premise of the controllers is that they have an opening through which the air flows. At higher pressures a flap or valve may throttle the flow by gradually closing the opening.

According to [14] three ranges of operation are distinguished in the flow controller characteristics, see figure 3.7:

Range 1: The pressure is low and the flow controller is fully open, flap or valve are in a fixed position. This part of the characteristic is simulated with the normal crack flow equation, which needs a flow coefficient c_q (volume flow at pressure difference 1 Pa) and an exponent n as parameters.

Range 2: The pressure is within the range where the position of the flap or valve can be adjusted, so that the flow is (roughly) kept constant. This part will be simulated, ideally, by a single constant flow rate, or with a curve approximated by a polynomial. The polynomial might give both flows increasing or decreasing with pressure. The transition point between range 1 and range 2 is simply defined by the intersection of the respective functions. In the same way, the transition point between range 2 and range 3 is defined by the intersection with the function for range 3.

Range 3: The pressure is beyond the range where the flow can be maintained at the level given by the function in range 2. The flap or valve may:

- ◇ close no further or have leaks, i.e. the flow is further increasing with pressure, or
- ◇ close faster, reducing the flow with increasing pressure to almost zero.

Range 3 starts at the intersection of the two curves in the ranges 2 and 3, respectively. Range 3 is also simulated with a polynomial. Care should be taken with steeply decreasing flows. If, due to this, the zone pressure rises if less air is supplied to the zone, or the opposite, then the network can no longer be solved. In reality, such flow controllers will probably also oscillate.

The polynomial approximation is used between the maximum pressures $p_{\mathrm{max,n}}$ and $p_{\mathrm{max,p}}$. As the polynomial approximation could oscillate at higher pressures which would lead to solver problem, a linear approximation beyond $p < p_{\mathrm{max,n}}$ and $p > p_{\mathrm{max,p}}$ is taken.

If the flow in range 3 is increasing the coefficients are:

$$\frac{\delta q}{\delta p} = q(p_{\max})/p_{\max} \quad \text{and} \quad q(0) = 0$$

In figure 3.7 the asymmetric flow controller `F4` shows this behaviour.

If the flow in range 3 is decreasing, the program will look for a root near $p_{\max,n}$ and / or $p_{\max,p}$ in the curve of range 3. In figure 3.7 the symmetric flow controller `F3` shows this behaviour. Make sure that $p_{\max,n}$ and $p_{\max,p}$ are a rough estimation of this root. Above this root the flow will remain zero.

Pollutants

Up to five pollutants can be simulated in COMIS.

Units for Pollutants. For each pollutant, separate units can be defined for concentration, sink strength and source strength. The definition of units is described under `&-PR-UNITS` on page 49. Under item "Header of data block" in section 3.1 the representation of units in the keyword desription and in the headers is explained.

```
&-NET-ZP zone-pollutants
```

Zone	Pollutant			
*ID	Initial Concentration	Source	Sink	Decay
(-)	(conc?)	[sour?]	[sink?]	[1/s]

Example
```
*z01        350.00
            1.00
```

In the above example, initial concentrations in zone z01 are defined. `(conc?)` means that the concentration unit of the pollutant in question, identified by its unique number, is used. If pollutant no 1 is 'CO2', and its concentration unit is 'ppm', pollutant no 2 is 'H2O', and its concentration unit is 'g/kg', then values for pollutant no 1 have to be input in ppm, and values for pollutant no 2 have to be input in g/kg though these values are input in the same column.

NOTE **Concentration Definition.** In COMIS, the concentration of a pollutant is defined as

$$c = \frac{\text{mass of pollutant in V}}{\text{mass of clean, dry air in V}}$$

or

$$c = \frac{\text{volume of pollutant in V}}{\text{mass of clean, dry air in V}}$$

This definition differs from the usual definition of pollutant concentration (mass of pollutant in relation to mass of pollutet, dry air). It is similar to the usual definition of humidity (mass of water vapour in relation to mass of dry air). However, if pollutant concentrations are low, the difference between the usual definition and the one used here is small.

Constituents of Air as Pollutants — Humidity as Pollutant. Air density in COMIS is calculated according to

$$\rho = \frac{\text{air mass + mass of water vapour + mass of pollutants in V}}{\text{total volume V}}$$

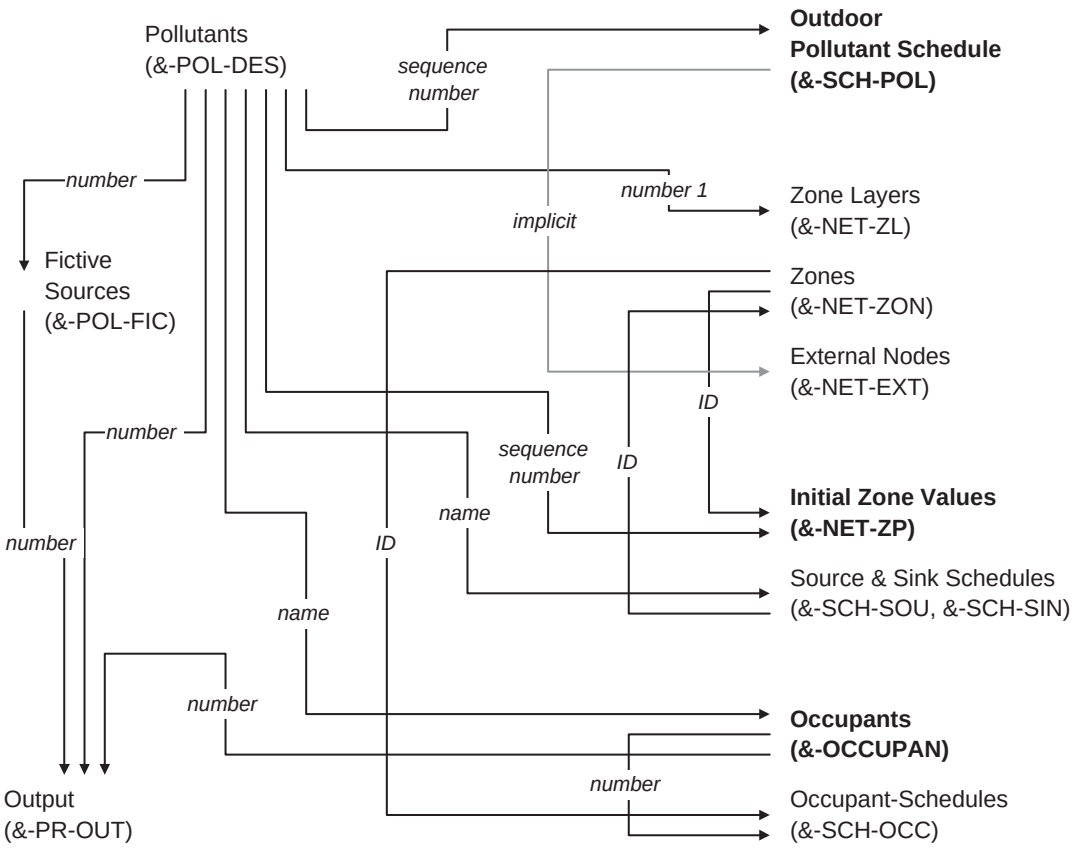


Figure 3.8: Keywords with definitions and references for the pollutant transport model. The arrows are directed from the defining to the referencing keyword. The labels show how an element is referenced.

ID:	identifier
name:	descriptive name
number:	explicit number
number 1:	layers only for first pollutant
sequence number:	same sequence as in definition
implicit:	automatically without explicit reference

A problem arises if a substance which is a usual constituent of air is treated as pollutant in COMIS. According to the above definition, such a substance would be counted twice when calculating the air density.

For humidity, a correct calculation can be achieved by

- ◇ setting the humidity to zero when water vapour is treated as pollutant,
- ◇ or by setting the pollutant concentration of water vapour to zero when the humidity is used.

For definition of these values, see `&-NET-ZON`, `&-NET-ZP` and respective schedules.

For carbon dioxide, there is no possibility to avoid double counting in the density calculation up to now. Normal concentrations of carbon dioxide in air are about 0.05 mass%, so the error is negligible.

Keywords Related to Pollutants. Figure 3.8 shows the relations between keywords relevant for the pollutant transport model. The arrows are directed from the defining to the referencing keyword. The labels show how an element is referenced. The keywords have the following tasks:

- ◇ `&-PR-UNIT`s (not shown in the figure)
  Units for the concentration, the source strength, and the sink strength of pollutants can be chosen.
- ◇ `&-NET-ZON` zones
  Schedules for source and/or sink strength of pollutants are referenced here
- ◇ `&-NET-ZL` zone layers
  Gradients for pollutant concentration are defined here
- ◇ `&-NET-ZP` zone pollutants
  Initial concentrations, source and sink strengths and absorption rates are defined here
- ◇ `&-SCH-SOU`, `&-SCH-SIN` source and sink schedules
  Time dependent source and sink strengths are defined here; the factors modify the initial source and sink strength input under `&-NET-ZP`.
- ◇ `&-SCH-POL` external pollutant schedules
  Time dependent external pollutant concentrations are defined here.
- ◇ `&-POL-DES` pollutant description
  Pollutants are defined here.
- ◇ `&-POL-FIC` fictive sources
  Fictive sources (see Effective flow rates) are defined here.
- ◇ `&-SCH-OCC` occupant schedule
  Occupant numbers and activity level factors for zones are defined here; without occupant schedule, no occupant will reside in any zone.
- ◇ `&-OCCUPAN` occupant description
  Occupant types and their contribution to pollutant sources are defined here.

Effective Flow Rates

Assume that the concentration $C_{n,k}(t)$ of pollutant no n in zone k would be a steady state concentration, i.e. $C_{n,k}(t) \equiv \text{const}$. If there is a fictive source with source strength $q_{fict,k}$ for that pollutant in zone k, and the concentration of the pollutant in the incoming air is zero, the total flow rate which is required to keep the concentration constant, is

$$q_{eff,k}(t) = \frac{q_{fict,k}}{C_{n,k}(t)}$$

This flow rate is called effective flow rate. However, the actual concentration usually is not constant, and thus for each time interval t usually a different effective flow rate results.

The effective flow rate is higher than the actual (fresh air) flow rate in a room if the concentration is below the steady state concentration associated with the fictive source. The effective flow rate is lower than the actual ventilation in the room if the incoming air passes other rooms in which the concentration is already raised. By definition the effective flow rate is the flow rate of fresh air that would keep the concentration in the zone at its actual value if the (net) source strength in the zone is given by the fictive source. Therefore it is the best value to look at when pollutant problems are to be solved. For energy loss/consumption it is better to look at the infiltration flow rate itself.

Effective flow rates are calculated according to the definition above. COMIS uses the fictive source $F_{\mathrm{m}}(n)$ and the concentration C of pollutant n in the zone under consideration, whenever the output of the effective flow rate `Fm` is demanded at `&-PR-OUTP`ut. The fictive, constant source strength is input at `&-POL-FIC`tive sources. Fictive sources can be defined in a number of ways for zones and for occupants.

The variables used here are:

F_{m}	Effective flow rate (at timestep t)
F_{Q}	Fictive Source
A_{F}	Floor Area of zone
A_{W}	Wall Area (all walls including ceiling and floor) of zone
V_{Z}	Volume of the Zone
$C(t)$	Concentration of the zone at a timestep t
$occ_num(t)$	Number of occupants of the zone at a timestep t
$< activity > (t)$	Mean activity of all ocupant types in the zone at a timestep t

For zones there are four ways to calculate the Effective flow rate controlled by the parameter 'Dependency indicator' in `&-POL-FIC`tive Sources:

1: absolute fictive source strength (Dep=0)

$$F_{\mathrm{m}} = \frac{F_{\mathrm{Q,1}} \cdot 1}{C(t)}$$

2: fictive source strength floor area dependent (Dep=1)

$$F_{\mathrm{m}} = \frac{F_{\mathrm{Q,F}} \cdot A_f}{C(t)}$$

3: fictive source strength wall area dependent (Dep=2)

$$F_{\mathrm{m}} = \frac{F_{\mathrm{Q,W}} \cdot A_w}{C(t)}$$

4: fictive source strength room volume dependent (Dep=3)

$$F_{\mathrm{m}} = \frac{F_{\mathrm{Q,V}} \cdot V_z}{C(t)}$$

For occupants there are two ways to calculate the effective flow rate controlled by the parameter 'Dependency indicator' in `&-POL-FIC`tive Sources:

1: fictive source strength proportional to the number of occupants (Dep=5)

$$F_{\mathrm{m}} = \frac{F_{\mathrm{Q,O}} \cdot occ_n um(t)}{C(t)}$$

2: fictive source strength proportional to the mean activity of all occupant types in the zone (Dep=6)

$$F_{\mathrm{m}} = \frac{F_{\mathrm{Q,A}} \cdot < activity > (t)}{C(t)}$$

The average occupant activity is the mean activity of all occupant types in the zone under consideration:

$$< activity >= \frac{\sum_{types} occ_num \cdot activity}{\sum_{types} occ_num}$$

Example Occupants produce CO2 dependent on their activity and metabolic rate, and move from zone to zone. The allowable concentration is set to 600 ppm above ambient concentration. The absolute CO2 level would then be 1000 . . . 1100 ppm.

The fictive source is set to 5.5E-6 m3/s CO2 which is about the average value for a person at light work. The room size dependency is set to 0 (fixed) in this case. An occupant enters a zone in which nobody has been before. The ventilation flow rate is adjusted for just one person in that zone. The CO2 concentration will start to rise from zero, leading to an infinite effective ventilation flow rate

at the beginning. The concentration will continue to rise to a steady state close to the allowable concentration. The effective flow rate will decrease accordingly.

If the effective flow rate is evaluated with a COMIS histogram output, the first effective flow rates will fall outside the classes of the histogram and will be counted in the class for overflow values. In fact this histogram shows the interesting part where the effective flow rate may become too low.

For sources that are more or less proportional to the floor area the dependency indicator is set to 1. Then the absolute value of the effective flow rate depends on the floor area of the zone. If the dependency indicator is set to 2, the source will be wall area (i.e. floor + ceiling + walls) dependent. With these settings sources from building materials or furniture can be judged. A similar possibility exists for volume dependent sources.

Histograms

With mode '-H*n*' the relative frequency distribution for zone, link and external node quantities can be output to histograms, see `&-PR-OUTPut` and `&-HISTO`.

Furthermore, histograms can be established for occupants. I.e., occupant type names defined under `&-SCH-OCCpant` may appear in the place of a zone name, if (and only if) the mode '-H*n*' is chosen for a zone output. If an occupant type is specified instead of a zone name, the histogram is established by evaluating the quantity denoted by the keyword in *all* zones where that occupant type resides. The frequency distribution is weighted by the number of occupants present in the respective zones. Thus the frequency distribution for occupants reflects the conditions an ensemble of occupants experiences. For example, concentrations in a crowded room have a high weight when evaluating the pollution load, concentrations in an empty room are disregarded for the evaluation.

The activity level of the occupants may also influence the severeness of an exposure, e.g. to pollutant loads. However, the evaluation is left to the COMIS user because there is no general model for taking into account the activity level.

Definition of Statistical Quantities in COMIS.

◇ Frequency distribution calculated in COMIS

For each time interval i of duration Δt_i, the COMIS histogram routine sorts the actual values in the respective classes. The duration of the time interval is used as weighting factor because time intervals may have different lengths. If no pollution routines are called, the time interval corresponds to the time interval between two scheduled events. If pollution routines are called, it corresponds to the internal time step of the pollution routines. In each internal time step of the pollution routines, the histogram subroutines are called.

◇ For time interval i

A certain quantity x has a value within class X in a zone or link k at a certain time interval i. This event is denoted by $\delta(x(k,i), X)$. During time interval i, the contribution to the frequency distribution h_k for zone, external node or link quantities is

$$h_{ik}(X) = \delta(x(k,i), X) \cdot \Delta t_i$$

The unit of $h_{ik}(X)$ is $[sec]$.

For a number of occupants per zone n_{occ} experiencing these conditions, the above value has to be multiplied with the number of occupants in zones with quantity x in class X, and summed up for all zones k,

$$h_i(X) = \sum_{\text{zones } k} n_{occ}(k,i) \cdot \delta(x(k,i), X) \cdot \Delta t_i$$

The unit of h is $[pers. \cdot sec.]$.

◇ For the whole evaluation period, the contribution of all intervals is summed up. The respective frequency distributions are

$$h_k(X) = \sum_i h_{ik}(X) = \sum_i \delta(x(k,i), X) \cdot \Delta t_i$$

for zones, external nodes or links, and

$$h(X) = \sum_i h_i(X) = \sum_{k,i} n_{occ}(k,i) \cdot \delta(x(k,i), X) \cdot \Delta t_i$$

for occupants.

◇ At the end of a simulation run, the sum over all classes is calculated for each histogram. These classes range from X_{min}, which is the 'below range' class, to X_{max}, which is the 'over range' class. This ensures that all events are counted.

The cumulated frequency H of class $X \geq X_{min}$ is

$$H_k(X) = \sum_{X'=X}^{X_{max}} h_k(X')$$

and

$$H(X) = \sum_{X'=X}^{X_{max}} h(X')$$

The total number of events (which is output by COMIS in the histogram summary) is

$$N_k = H_k(X_{min})$$

for zones, external nodes or links, and

$$N = H(X_{min})$$

for occupants.

Depending on the number of occupants and on the share of the total simulation time, during which zones are occupied, N_k and N may be different.

The units of H_k and N_k are $[sec.]$. The units of H and N are $[pers. \cdot sec.]$.

◇ The relative frequency distribution is the frequency distribution normalised with the total number of events. The relative frequency distribution is output by COMIS together with the total number of events and some further values. The output is explained in more detail for the factory example on pages 133ff.

$$f_k(X) = \frac{h_k(X)}{N_k}$$

for zones, external nodes or links, and

$$f(X) = \frac{h(X)}{N}$$

for occupants.

With the histograms output by COMIS, further values can be calculated outside COMIS.

Histograms and quantities derived from histograms for occupants can in principle also be constructed from the time series of the quantities for each zone and the time series of number of occupants for each zone. Differences to COMIS histogram output are negligible if the time steps are small enough.

Due to the shorter internal timestep used inside the COMIS pollution transport routines (see page 42) COMIS histograms can give detailed statistics without increasing the amount of simulation time steps and output.

An example of occupant-weighted frequency distributions can be found on page 133.

Occupants as Source and Occupants as Sensor

The occupant-weighted frequency distribution uses the presence of an occupant type in a zone. Every zone quantity may be evaluated with an occupant-weighted frequency distribution.

If the second data block under `&-OCCUPAN` (pollutant name and source strength) is omitted, or if the activity level in `&-SCH-OCC` is set to zero, the respective occupant type can still be used for establishing occupant-weighted frequency distributions. However, in that case the occupant type will not be the source of any pollutant. I.e., the occupant acts as a sensor.

If there is a second data block under `&-OCCUPAN` (pollutant name and source strength), and if the activity level in `&-SCH-OCC` is not zero, then the occupant type contributes to the pollutant source strengths in zones where the occupant number is not zero. In that case, the occupant acts as source – and eventually, as sensor, too.

Default Values

COMIS has default values for most variables read from the input file. Others are initialized with '0'. Most defaults are assigned in the main program. For specific values, see column 'Default value' in the tables following the data section keywords. From each input line that contains data, COMIS reads a fixed number of variables. As soon as there is no more data in the line, and the program still tries to read data, the variables are filled with their defaults.

COMIS can report defaults it uses when reading an input file. The Keyword to echo defaults is the 'DEFAULT' under the `&-PR-SIMU`lation options. The echo is sent to the COF-file.

For example, the default input echo might look like this:

```
Def. value  0.000    used for Ref height zone Zz in line:  E01  Examp_Zone    25
```

The first part states that a default of 0.0 has been used. The second part is the line from the CIF-file, in which this default has been inserted.

Schedules, Time Steps and Simulation Dynamics

Names of schedules which are activated by referencing them from another data section have to start with a predefined character indicating which component is affected by the schedule. Example: names of schedules defining opening factors for windows must start with 'W'.

Schedules allow changing certain parameters in time, e.g., weather data. Schedules give a point of time for an event. The scheduled quantity takes a new value at that point of time, and from then on remains constant until a new event (new schedule entry) sets a new value. Which values are taken for scheduled quantities depending on the time covered by the schedule is shown in figure 3.9.

If the schedule has no event at the start time of the simulation, COMIS will use the value at the last event before the start time. If the schedule starts with an event after the start time, the scheduled values are set to the defaults provided by COMIS or to primary values defined in the input file.

All network quantities (especially flows and pressures) keep the values that have been found from solving the network at a point in time during the following interval. The only exception are concentrations of pollutants for which the interval is subdivided according to the time constant of the network. That means, airflow calculation is a steady state simulation and the pollutant transport is calculated with a dynamic model.

Schedule events trigger the next calculation of the network solving. The number of time steps in a COMIS simulation is determined by the number of schedule events with unique points in time. As mentioned above pollutant concentrations can change their values during an interval. Therefore it could be interesting to force additional concentration output between schedule events. This can be done by setting the `POLSTEP` option under `&-PR-SIMU`lation. At this additional points in time network solving will not be recalculated. The results may differ from those obtained by forcing output with schedule events without changing the scheduled boundary conditions. In the latter case, both network solving and pollutant calculation are forced. While driving forces remain unchanged if only the pollutant calculations are performed, changing driving forces may result from changing air density due to different pollutant concentrations if network solving is invoked.

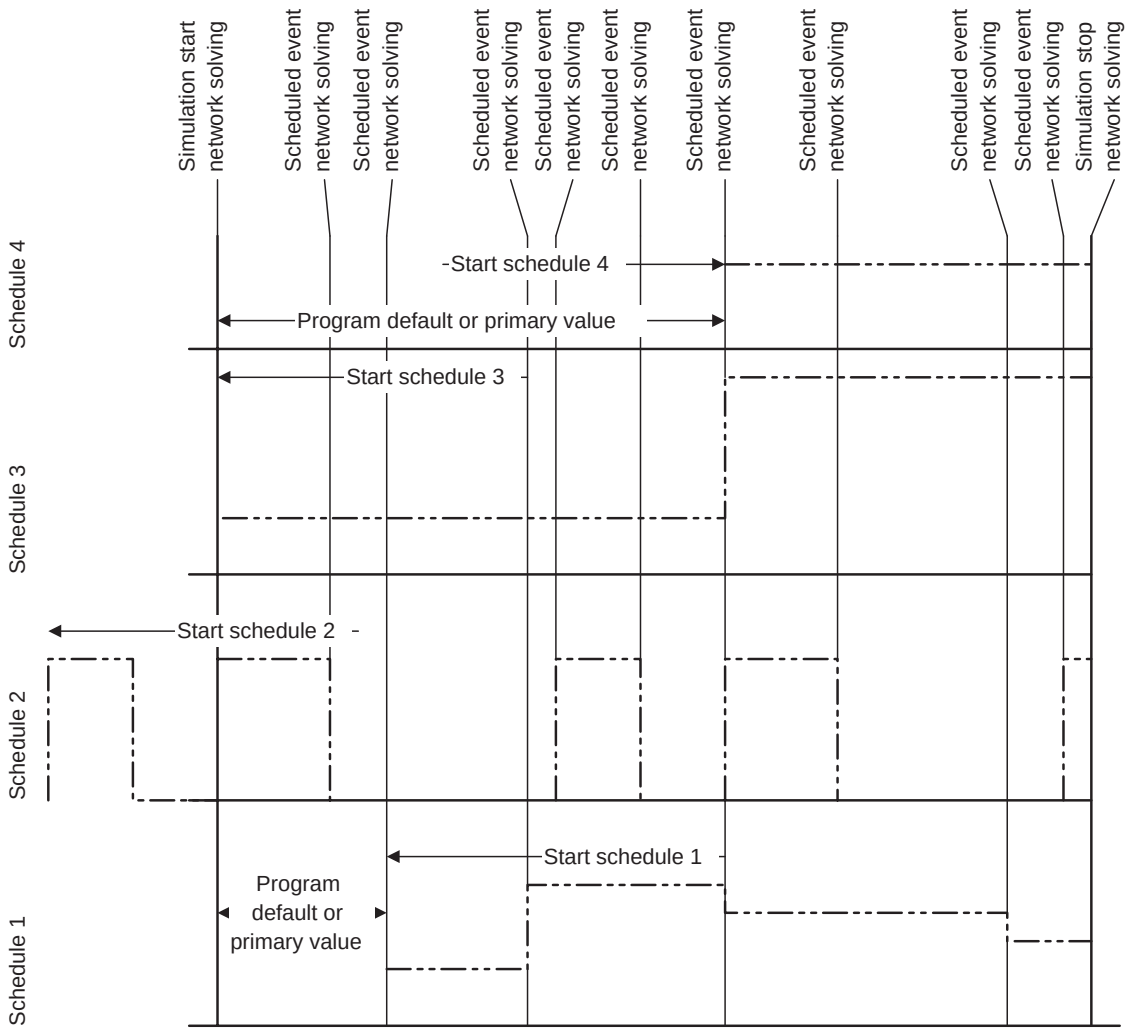


Figure 3.9: Schedules

Notes on Convergence

Definition: The overall resistance of a node is the replacement resistance of all links to the node.

Definition: Negative-Nodes are nodes for which:

- ◇ more air flows out of the node if the pressure is lowered,
- ◇ less air flows out (or more in) if the pressure is raised.

Definition: Normal nodes with passive resistances have the reverse:

- ◇ more air flows out if the node pressure is made higher,
- ◇ more air flows in if the node pressure is made lower.

The use of the Air flow components fan `FA`, flow controller `F1` through `F4`, related flow component `RF` and test data `TD` may lead to unstable networks. The overall resistance of a node may become negative if one of these components is used.

All nodes with an overall negative resistance lead to a situation where the network cannot be solved. The zone pressure cannot converge.

Tests for negative-nodes could be done, but zones can become negative-nodes depending on the pressures in the network. At some pressure distribution the node becomes locally negative. As the solver jumps pretty irregular through the pressures, this problem can occur in some cases in the numerical solution while it would not exist in reality.

However, negative nodes almost never exist in reality. The few exeptions are:

1: an ill-conditioned control loop for a fan with positive feedback (this will lead to vacuum or the maximum pressure the fan can blow into the node).

2: the detonation of an explosive (chemical reaction speeds up as the pressure rises, making the thing unstable)

3: a loose hanging flap/part in a wall closing at a certain pressure

4: a flow controller (it does the same as a loose hanging flap/part)

The message COMIS gives is:

```
MATRIX SINGULAR in ZONE \# nn
```

The number nn can be the negative node at either side of the negative-link.

The characteristics of the above mentioned components can be (locally) negative (flow over the component, in the direction of the pressure vector, drops as pressure rises). Depending on the other links and points of operation on their characteristic a negative-node could result.

COMIS has some added network checks which prevent most of the 'MATRIX SINGULAR' conditions. If one gets this message and fans, flow controller test data or related flow components are in the network, one should try first to replace these components step by step with a standard crack component. This will probably locate the cause of the negative node.

For fans negative parts in the curve do exist only for ill-conditioned polynomials. The fan in COMIS is described as flow as a function of pressure difference. This should not be confused with the pressure bump (contraflexure) seen in fans with forward bend scoops. One cannot input such a curve in COMIS!

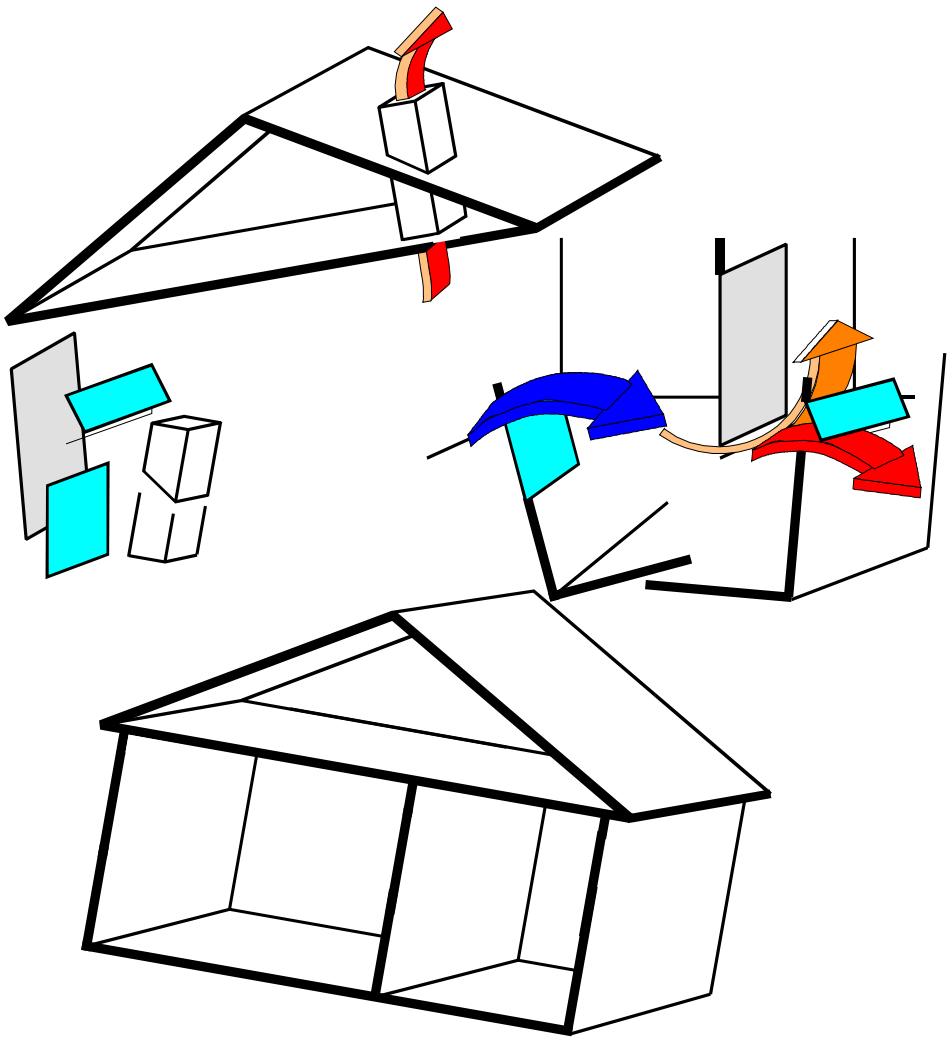
At flow controllers one should watch for the part where the flow decreases if pressure increases. If this decreasing flow rate is not compensated by an increasing flow rate in other links in the zone in which the flow controller is placed, then the node becomes negative.

Related flow components that have their reference link connected to the same node, and have a flow direction opposite to the flow of the reference link, and have a larger flow rate than the reference

link form a negative link. If this is not compensated by (large enough) other links, the node will get negative.

Related flow components in zones further away from their reference link may cause a series of nodes to become negative, but only if their flow rate is (much) larger and opposite to the flow in the reference link. If a large exchange flow rate must be provided by a related flow component, then one should use two related flow links with the same RF type, but link them in opposite directions to the node. This will never cause convergence trouble.

4 Data Sections



In this chapter all input parameters are described in detail. The description of the parameters is structured according to data groups:

- Problem description
- Network description
  - Air flow components, Zones, Links
- Schedules
- Cp-Values
- Environment description
- Pollutant description
- Occupant description

It is recommended to follow this sequence of data groups and sections when establishing an input file. If another data section is required before a certain keyword may occur, this is stated in the description of the respective keyword.

The following sections will show the headers, some example inputs, and explanations of the input. The inputs are shown in typewriter characters.

4.1 Project Description

&-PR-IDENtification (Project Identification)

```
&-PR-IDENtification
  ________________
 |1.|Problem Name |
 |__|             |
 |_________________|
  ________________
 |2.|Version Name |
 |__|             |
 |_________________|
```

Example
```
This is an example of the project name.
You may write text here to describe the project case.
The maximum number of lines is 10.
 |
24-June-89
```

NOTE At least one header line, i.e. one line with first non-blank character '|' must be between description of problem and version name. Otherwise version name is not recognized and the contents of the line ist appended to problem name.

Description:

Parameter	Description	Default value	Input format, range
Problem Name	short description of project / simulation model	' '	String 800 char (10 lines with max. 79 characters + crlf each)
Version Name	version name, date, etc.	' '	String 160 char

&-PR-UNITS (Project Units)

```
&-PR-UNITS

  -------------------------------
 | Unit Conversion Definitions   |
 | Name    |  Input   |  Output  |
 |_________|__________|__________|
```

Example 1

```
 pres        Pa        Pa-OSR
 temp        C         F
 humi        g/kg      g/kg
 INPUT
 cflo           m3/s
 prof           alpha
 OUTPUT
 mean           s
```

Example 2

```
 sour_co2    kg/s      mg/s
 sour_h2o    kg/s      mg/s
 sour_odour  olf       olf
 conc        ppm     ppm
 conc        g/kg    g/kg
 conc        decipol decipol
```

If the keyword INPUT is seen, the units in the following lines are only used for input. Conversely, if the word OUTPUT is seen, the units are used only for output.

If neither INPUT nor OUTPUT is seen, the first value following the unit identifier is used for input and the second for output.

The wind velocity profile "unit" is available for input only (no profile output).

The air change rate (ach), mean age of air (meanage) and energy units are only available for output and must follow the OUTPUT keyword. All the other units may be used for both input and output conversions.

The data section `&-PR-UNITS` can be used more than once. This allows having different units in different data sections within one input file. The units declared by `&-PR-UNITS` are valid for all data sections following these declarations until the end of the input file or until another unit declaration is encountered that changes the units defined explicitly before.

Description:

Parameter	Description	Default value	Input format, range
Name	identifier for physical quantity; also used as reference in the headers to identify the corresponding quantities; or: keyword `INPUT` or `OUTPUT`, if only one unit is given	—	string element from table below, 4 significant characters; or `INPUT` or `OUTPUT`
Input	unit for input values	see table	string element from table
Output	unit for output values	see table	string element from table

However, it is recommended to use the unit conversion only once per input file. It is also advisable to place the data section `&-PR-UNITS` as early as possible.

The `&-PR-UNITS` input is the preferable method of defining the unit conversions because all information on the simulation situation is fully contained in the input file. It is not recommended to use the COMIS.SET file to define the units (see also section 2.4 and apppendix C). If units are set in the SET-file, it is impossible to tell which input units were used unless this file is kept with the project files, too.

The values of the quantities listed in the table below can be converted from input units to COMIS internal units, or from COMIS internal units to output units. The internal units are at the same time the default units for input and output.

Name	Quantity	Default Unit	Other possibilities
AIRLeakage	Air Leakage	Cm(kg/s@1Pa)	kg/h@1Pa, kg/s@10Pa, kg/h@10Pa, dm3/s@1Pa, m3/s@1Pa, m3/h@1Pa, dm3/s@10Pa, m3/s@10Pa, m3/h@10Pa, ELA4, ELA10
FLOW	Air Flow	kg/s	g/s, g/h, kg/h, m3/s, dm3/s, dm3/h, m3/h
PRESsure	Ventilation Pressure	Pa	mmH2O, mmHg, inH2O, inHg, hPa, kPa option -OSR to each unit
TEMPerature	Temperature	C	K, F, Ra, Re
HUMIdity	Humidity	g/kg	kg/kg, mass%, %rh
SOURce	Pollutant Source	kg/s	g/s, g/h, kg/h, mg/s, ug/s [a], olf [b]
SINK	Pollutant Sink	kg/s	g/s, g/h, kg/h, mg/s, ug/s [a], olf [b]
CONCentration	Pollut. Concentration	kg/kg	g/kg, mg/kg, ug/kg, m3/kg, dm3/kg, ml/kg, ul/kg, mass%, ppm, ppb, part (any rational combination of g, kg, mg, ug, m3, dm3, ml, and ul are possible, e.g. mg/g, dm3/mg) [a], decipol [b]
VELOcity	Wind Speed	m/s	cm/s, kt, ft/min, km/h, mile/h
Input option only			
CFLOw	Component Flow	m3/s	g/s, g/h, kg/s, kg/h, dm3/s, m3/h, dm3/h
PROFile	Profile	alpha	alpha, z0
Output option only			
ACHAnge	Air Change Rate	1/h	1/s
MEANage	Mean Age of Air	h	s
ENERgy	Energy	KWh	J

[a] u means μ

[b] units for the special pollutant named `ODOUR`

NOTE: In the headers all quantities are given with the corresponding unit identifiers to indicate the used input units. For quantities which have only one valid unit, the prescribed unit is given explicitly in the corresponding header. For example, input values for barometric pressure always have to be in kPa. There is also no choice for length input. Depending on the quantity, the unit is always m or mm.

NOTE: There is a option '-OSR' which may be appended to the pressure unit (see example). With this option, the outside stack pressure is taken as pressure reference. For further details, see notes on pressure reference on page 33.

NOTE: Different units for source, sink and concentration may be defined for each pollutant. This is done by entering multiple lines for `sour`, `sink` and `conc` as shown in example 2. As only four characters of the name are used to identify the physical quantity, the name may be longer e.g. to indicate for which pollutant the unit is used. The first unit definition will be used for pollutant number 1, the second for pollutant number 2 and so on. If more than one pollutant unit is entered, but fewer than the number of pollutants defined, then the last line will be used for the unit conversion for

the remaining pollutants. If only one line is entered, the same units will be used for all pollutants (so COMIS 3.1 is backwardly compatible with older CIF-files).

NOTE: The units 'olf' for source strength and 'decipol' for concentration must be used for the pollutant 'ODOUR'. They must be defined for input and output. They must not be used for other pollutants.

&-PR-SIMU lation (Project Simulation Options)

```
&-PR-SIMUlation options
 ------------------------------------------------------------
| Simulation Option Keywords: One keyword per line           |
|                             Keywords may be preceded by NO |
|____________________________________________________________|
| VENTilation                         CHECKCondensation      |
| 2VENT                               INPUT                  |
| POLutants <no>                      DEFAULT <no>           |
| POLSTEP <step> [<time> <step2>]     SET                    |
|                                     UNIT                   |
|                                     ECHOSCHedule           |
|                                     ONSCREEN [time|progress |
| SCHEDtime <time>                       |niter|pol<n>| error]|
| HISTOtime <time>                    STEADY                 |
| STARTtime <time> [CONT|REUSE]       DEBUG <no>             |
| STOPtime  <time> [KEEP]             LOOPRHO                |
|____________________________________________________________|
```

Example
```
VENTilation
POLlut 2
DEFAULT 1
POLSTEP  60    7200   600
STOPtime  1988jun01_11:00 keep
```

Each keyword may be preceded by the word 'NO' thus canceling the keyword. Only one keyword per input line is allowed. Keywords may be given in any order.

It is possible to have a start time, which is equal to the stop time. That means only one time-step is performed. It is not allowed to have a stop time earlier than the start time. If no times are given, the program will search the meteo schedule (see `&-SCH-METeo` section) for a time span. This will only work correctly, if the full time description (see section 3.3) is being used. Therefore, start and stop time of the simulation should be defined under the keyword `&-PR-SIMU` options.

All schedules will be searched for the event at or before simulation start time. If no value is found, default values will be taken for the first time step.

A third keyword handled by the time management system (TMS) is the keyword SCHED, which specifies the time the program starts to read and process the schedules. The schedule time may be different from the start time to allow some fixed pre-conditioning and delay the normal operation of the schedules, and is equal to the start time as a default. The SCHED keyword can be important, e.g., if one wants to split a run over one year into 4 runs simulating three months each. If one wants that the second run starts with the same conditions the first run ends, a schedule time equal to the start time of the first run can be specified.

In order to save some time the TMS file (see COMIS file structure on page 20) of the last run can be reused again. This saves time that would be necessary for creating and sorting the schedule data.

If one wants to keep the TMS file of the current simulation run for later use, the KEEP option must be used at the end of the stop time definition.

To tell the program that the TMS file from a former program run shall be used the CONT keyword or the REUSE keyword must be specified behind the start time. In case the CONT keyword is used, the start time of the following run must be the same as the stop time of the preceding run for which the TMS file was created and kept for later use . If the REUSE keyword is used, the whole TMS file from the last run is used again. This option must be used carefully. It does not work if a line in the input file or in a schedule file has been inserted or deleted. Furthermore, start

and stop time have to be the same as in the previous run. The values of schedule-data or other

Description:	
Parameter	Description[a]
VENTilation	Calculation with COMIS, ventilation results written to COF-file (concise format)
2VENT	Calculation with COMIS, ventilation results written to COF-file (detailed format)
POLlutants [<No>]	pollutant routines are called; if a No is specified, only pollutants up to number <No> (defined under &-POL-DES) are active; if no number is specified, the maximum number of pollutants (5) is made active
POLSTEP <step> [<time> <step2>]	<step> is the timestep (in seconds) for pollutant calculations *until* <time> (in seconds), then <step2> (in seconds) is used as pollutant calculation timestep for remainder of simulation
CHECKCondensation	Warns if condensation occurs (i.e. if air relative humidity reaches 100%; does not evaluate condensation due to low surface temperatures)
INPUT	Input (CIF-) file is copied to COF-file
DEFAULTs [<No>]	Lack of data and d[e[f[a[u[l[t]]]]]] values are specified in COF-file; provides three levels (<No> = 1..3): 1 = Echo *all* defaults (Default option) 2 = Echo *only* defaults used whenever "d[e[f[a[u[l[t]]]]]]" appears in the CIF-file 3 = Echo *only* defaults used whenever there is no more data at the end of input lines
SET	SET-file is copied to COF-file
UNIT	Implemented keywords are summarized in COF-file (be aware that this list has not been updated!)
ECHOSCHedule	Echo all changes in schedules in COF-file
ONSCREEN [TIME | | PROGRESS | NITER | | POL<No> | ERROR]	Output scheduled time or simulation progress, number of iterations, concentration of pollutant No, or convergence error in mass flow on screen
STEADY	Calculate steady state concentrations for all time steps
DEBUG <no>	Echo the temporary time management files to COF-file; <no> = 1 echo COMIS.DAF; <no> = 2 echo COMIS.DAF; <no> = 3 echo COMIS.DAF and COMIS.TMS
LOOPRHO	Recalculate air density in zones during the network pressure iteration loop
SCHEDtime <time>	Start time of processing the schedules <time> is the date and time in date_time string format (see section 3, page 30)
HISTOtime <time>	Start time of processing data for histograms (if not given, simulation period is assumed) <time> is the date and time in date_time string format (see section 3, page 30)
STARTtime <time> [CONT | REUSE]	Start time of simulation <time> is the date and time in date_time string format (see section 3, page 30) CONT after the start time means an old TMS-file has been kept and shall be reused, with the start time of the new run equal to the stop time of the previous run. REUSE stands for using the whole TMS-file of the last run again, with the same start and stop time and unchanged length of the input file(s).
STOPtime <time> [KEEP]	End time of simulation <time> is the date and time in date_time string format (see section 3, page 30) KEEP after the stop time means that the TMS-file shall be kept after the simulation is completed.
	"1900" is used if the year is missing and "1900jan01" if the date is missing in the date_time string for <time>. "1900jan01_00:00:00" is used if no date_time string or "d[e[f[a[u[l[t]]]]]]" is specified.

[a] for the meaning of the file names, please refer to section 2.4

input values, however, can change as well as the weather file.

The keyword POLSTEP forces pollutant transport solving with predefined time steps, independent of scheduled events. In the example given above, the pollutant transport is evaluated every minute ($< step >= 60$), but 2 hours after starttime, ($< time >= 7200$) the time step is switched to 10 minutes ($< step2 >= 600$)

It is suggested to force recalculation of the zone air density during iteration by setting the keyword LOOPRHO.

&-PR-OUTPut (Project Output Options)

```
&-PR-OUTPut options
```

Output Options Keywords: One keyword per line		
Format:		
Keyword-mode [name[,name][,name-name][,...]]		
Keyword-mode [name[,name][,name-name][,...]]		
mode: S : Storing each value in *.CSO file		
T : mean value of Total simulation period in *.COF		
Hn: Histogram of Histo-period in *.CHO		
name: corresponding to the keyword, name is a zone, a link or an external node name		
Keyword Parameter	**Keyword Parameter**	**Keyword Parameter**
per Zone	per Link	Ambient conditions
FZ Flow rate	FL Flow	WA Wind velocity
PZ Pressure	TL Temperature	TA Temperature
TZ Temperature	VL actual Value	HA Humidity
HZ Humidity	PL Pressure	
IZ Infiltration		
AZ outdoor Air change rate	per External node	per Building
	PE Pressure	FB Flow matrix
MZ Mean age		FW Flow matrix W
EZ air change index		IB Infiltration
		AB outdoor Air change rate
On number of occupants		MB arithmetic Mean age
Yn activity factor of occ type n		RB disparity of mean age
per Zone and Contaminant		NB Nominal time constant
Cn concentration	Examples:	EB air change Efficiency
Sn sink	FZ-S zn1,zn2-zn7,zn10	LB ventilation heatloss
Qn source	FL-T li1,>li3,<li3	
Fn eff flow	> := flow From --> To	
	< := flow From <-- To	

Example

```
PZ-S  1-3,38,100-115,200
PZ-H2 1-3
PZ-T  1-5,4-7,6-10,10,38,100,101
PZ-T  200
FL-S  >L01, <L01, >L12, <L12
TZ-S  z1-z20
TZ-T  zon3-zon5,kit,liv
TZ-H2 zon3-zon5,kit,liv
TA-S
C1-T  1,2,3
C1-H1 1,2,3
C1-H1 occ1, occ2, occ3
```

The general syntax of the output definition is

`keyword-mode namelist`

Keywords are used for creating output files (CSO- and CHO-files) that may later be used by a spreadsheet program, and for summary output to the COF-file. They consist of two letters or

Description:	
Parameter	Description
keywords with namelist	
FZ	Flow per Zone
PZ	Pressure per Zone
TZ	Air Temperature per Zone
HZ	Humidity per Zone
IZ	Outdoor air infiltration per Zone
AZ	Outdoor air change rate
MZ	Room mean age of air per Zone
EZ	Room air change index per Zone
On	Number of occupant type n per Zone, $1 \leq n \leq 5$
Yn	Activity level factor of occupant type n per Zone, $1 \leq n \leq 5$
Cn	Concentration of Contaminant[a] n per Zone or External node, $1 \leq n \leq 5$
Sn	Pollutant Sink of Contaminant[a] n per Zone, $1 \leq n \leq 5$
Qn	Pollutant Source Strength of Contaminant[a] n per Zone, $1 \leq n \leq 5$
Fn	Effective Flow calculated with Fictive source n per Zone, $1 \leq n \leq 5$
FL	Mass Flow per Link; link name only: net flow rate, < before link name: flow 'To to From', > before link name: flow 'From to To'.
TL	Temperature per Link
VL	"Actual value" per Link in accordance with the respective link schedule (for fan or window)
PL	Pressure per Link
PE	Wind pressure per external node
keywords without namelist	
WA	Wind Velocity
TA	Outdoor Air Temperature
HA	Outdoor Humidity
FB	Flow matrix for the building
FW	Flow matrix for the building including external nodes
IB	Outdoor Air Infiltration for the building
AB	Outdoor Air Change Rate for the building
MB	arithmetic mean of building mean age of air
RB	Root Mean Square (RMS) disparity of zones mean age of air with respect to building mean age of air
NB	Nominal time constant of building mean age of air
EB	Air Change Efficiency of building
LB	Ventilation Heat Loss Energy of building
modes	
-S	Output to CSO-file for each timestep
-T	Output of mean values for simulation period to COF-file
-Hn	Output of histograms for histogram period to CHO-file

[a] 'Contaminant' and 'pollutant' are used synonymously

one letter and one digit. The first character denotes the physical quantity. The second describes the component for which the quantity is to be output. If a digit is required, the component is a pollutant or an occupant type in a zone. The range is $n = 1 \ldots 5$, as up to five pollutants or occupants may be defined (see sections 4.6 on page 123 and 4.7 on page 125).

A keyword is followed by a mode indicator. In mode '-S' actual values of the timestep are stored (i.e. sent to the CSO-file). Mode '-H*n*' produces a histogram of type n for the histogram period (default: simulation period) in the CHO-file. Histogram types are defined with a separate keyword `&-HISTO`, see page 61. The file names are composed of the keyword and the root name of the project (see section 2.4, page 21), and the extension is `.cso` in '-S' mode or `.cho` in '-H*n*' mode, respectively. In '-H*n*', *n* is the type number of the histogram to be used. It must be defined under section `&-HISTO`. Mode '-T' produces a table of mean values for the whole simulation period in the COF-file.

NOTE The '-H*n*' option works only if the element covered by the keyword allows a further specification, i.e. for zones, links and external nodes (such as the list of zones in `PZ-H2 1-3`, i.e. 'Store Pressure for Zones 1 to 3').

A namelist must follow the keywords for zones and links. Building and ambient output per definition have only one element, therefore no namelist specification is necessary. A namelist consists of single element names or ranges of element names, separated by commas or spaces. Ranges are two element names separated by a dash, e.g. `zon3-zon5`. In this example it is assumed that `zon3` is defined before `zon5` under `&-NET-ZON`es (see page 92). Then output will be produced for all zones from `zon3` to `zon5` in the list of zone definitions.

For example, `PZ-S 1-3` would output the zone pressure for zones 1 to 3, and `TA-S` would output the ambient temperature. Note that the definition of 'Ambient Temperature' ('TA') does not have any parameters, since there is only one element 'outdoor conditions'.

Occupant type names defined under `&-SCH-OCC`pant may appear in the place of a zone name, if (and only if) the mode '-H*n*' is chosen for a zone output. If an occupant type is specified instead of a zone name, the histogram is established by evaluating the quantity denoted by the keyword in *all* zones where that occupant type resides. The frequency distribution is weighted by the number of occupants present in the respective zones. For the use of histograms see details on COMIS histograms on page 40) and an example for occupant-weighted frequency distributions on page 133).

NOTE However, ranges of occupant type names are *not* allowed. If output for occupant types 1 to 3 is desired, use e.g. `C1-H1 occ1, occ2, occ3`.

NOTE For the output keyword 'C*n*', names of external nodes may be given. Thus ambient pollution concentration can be output. If the same name is used for a zone and an external node, the zone concentration is output.

NOTE The `FL-...` option, mass flow per link, normally gives the net flow rate. To get the flow separately for each direction, put a '<' in front of the link name for flow 'To to From', and a '>' in front of the link name for flow 'From to To'.

NOTE The flow matrix with the mean values is written to the CSO-file and not into the COF-file. Therefore, to get the flow matrix with mean values, the `FB-S` option has to be specified in addition to `FB-T`.

The definition of temperature output with `TZ-S z1-z20` as given in the example means that the values for zones z1 to z20 are stored in the file `<name>-TZ.CSO`. The `TZ-T ...` makes COMIS write the mean values in the COF-file. The specification `C1-S 1,2,3` as given in the example means "create output in a CSO-file for concentration of gas number 1 in zones 1, 2 and 3".

NOTE The definitions of the output produced by the building output options, keywords `xB` and `FW`, are described in more detail in [9].

&-PR-CONTrol (Project Control Parameters)

```
&-PR-CONTrol parameters
```

1.	Under Relaxation Factor [-]	Tolerances: absolute EpsFA [flow]	Tolerances: Relative EpsFR [-]	Tolerances: EpsCJ [flow]	Start Number of Iterations [-]	Link Flow Pressure Laminar Flow DifLim [pres]

2.	Pressure initialization frst time step	Pressure initialization later time steps	Max Iterations allowed	Solver selector (OPTIONAL)
				1=Newton (with given Relax)
	0=zero Pressure	0=zero Pressure		3=Walton Steffensen
	1=linear system	1=linear system		5=Walton 2 fixed relax.fact
		2=previous value		6=trust region line search
	stp1Init [-]	stp2Init [-]	Miter [-]	slvSel [-]

Example

```
  1          1.E-6       1.E-4         d          1       1.0E-4
  1                      2                 50             6
```

Link flows are approximated by a linear function at pressures smaller than DifLim. This is effective to avoid problems with large openings.

Values mean the same thing for both initialization parameters in data block 2. While `stp1Init` applies to the first time step of the simulation, `stp2Init` applies to all subsequent time steps.

The meaning of the values is:

0: Zero pressures, accounting only for hydrostatic variations, before starting solution at the indicated time step. Initial zone densities are based on zero pressures.

1: Find initial pressures by solving a semi-linearized version of the system ("semi-" because not all flow elements define a linearized model). The solver takes one step on the semi-linearized system. It does not attempt to solve the mix of linearized and nonlinearized models. Initial zone densities are based on zero pressures.

2: Take pressures calculated at previous step as initial guess for pressures at this step. Initial zone densities are based on the pressures found at the last step.

By default, the zone density remains unchanged throughout the solution at each time step. To make the solver update zone densities as it updates zone pressures, add the line `LOOPRHO` to the `&-PR-SIMU`lation options section (see page 52).

The fourth parameter in data block 2, can take the values `slvSel` = 1, 3, 5, and 6. Solver 6 (default) should be slightly more reliable than solver 5. In some cases it will take fewer iterations at each time step, and in other cases it will require more iterations. A description of solver 6 and a report on tests with solver 5 and 6 can be found in reference [23].

In all cases, solver 6 should converge whenever solver 5 can converge, and it should also converge in many cases that caused solver 5 to fail (when using duct T-junctions, all bets are off). When using `slvSel` = 5, you should probably set `stp1Init`=0 in order to minimize the number of iterations taken at the first time step. On the other hand, slvSel=6 works better with `stp1Init`=1. Both solvers should take many fewer iterations at subsequent iterations when using `stp2Init`=2, as opposed to 0 or 1. When using solver 6, the underrelaxation control parameter (first parameter of data block 1) has no effect.

Recommended parameter values for solvers number 5 and 6 are summarized in the following table:

slvSel	underrelaxation factor	stp1Init	stp2Init
5	0.5 . . . 1.0	0	2
6	no effect	1	2

Description:

Parameter	Description	Default value	Input format, range
Under Relaxation Factor	Newton pressure corrections are multiplied with this factor if pure Newton iteration steps are used; practical range Relax=0.5, . . . , 1.0	1.0	
Tolerances	Stop criteria for iteration		
Absolute EpsFA	Absolute tolerance for mass flow balance per flow	1.0E-6 kg/s	
Relative EpsFR	Relative tolerance for mass flow balance per flow (100% is total flow through the zone)	1.0E-4	
EpsCJ	Tolerance for CORR*JAC(i,i), i.e. the Newton pressure correction multiplied with the derivative $d\left(\sum_{\text{link}}(flow)\right)/dp_z$. This is an estimation of the error in the sum of the flow per zone.	0.2*EpsFA	
Start Number of Iterations	The number of pure Newton iteration steps before switching over to optimum relaxation	1	
Link Flow Pressures Laminar Flow DifLim	Link flows are approximated by a linear function at pressures smaller than DifLim.	1.0E-4 Pa	
Pressure Initialization frst time step	Pressure initialization in the first time step of the simulation stp1Init=0: Zero pressures before starting solution; initial zone densities are based on zero pressures stp1Init=1: Find initial pressures by solving a semi-linearized version of the system.	1	Integer 0 . . . 1
Pressure Initialization later time steps	Pressure initialization for all but the first time step of the simulation stp2Init=0,1: see stp1Init=0,1 stp2Init=2: Take pressures calculated at previous step as initial guess for pressures at this step: initial zone densities are based on the pressures found at the last step	2	Integer 0 . . . 2
Max Iterations allowed	If a solution is not found before the given number of allowed iteration steps, the program breaks and reports that the output for this time step may be wrong	50	Integer
Solver selector (OPTIONAL)	1 = Newton (with given Relaxation) 3 = Walton Steffensen 5 = Walton 2 fixed relaxation factor 6 = trust region line search	6	Integer 1, 3, 5, 6

Solver 5: Walton 2 fixed relaxation factor

The solver first starts with a number of pure Newton iterations using the under-relaxation factor (data block 1 data 1). The start number of iterations is given as user input under `&-PR-CONT` data block 1 data 5. Useful values are 2 or 1.

After those iterations the Walton solver[5] is used. It makes use of the air flow rate and the derivative d/dP of the flow per air flow component. This is a very simple solver that only switches the relaxation factor, depending on the convergence, but the performance is very good.

convergence	relaxation factor
$\leq$0.3	1.0
0.3 to 0.7	0.75
$>$ 0.7	0.5

[5] The Solver has been developed by George Walton of NIST for the AIRNET program

&-HISTO gram (Histogram Definition)

```
&-HISTO
  ------------------------------------------------------
 | Histogram | Number of | Lowest Class| Highest Class|
 |           |  Classes  |             |              |
 |   (-)     |    (-)    |    (?)      |     (?)      |
 |___________|___________|_____________|______________|
```

Example

```
      3           10         0.0           1
      1           21         0.0           10.E-3
      2           6          -50.0         50.0
```

&-HISTO defines histograms to be used in &-PR-OUTPut section. Histograms are identified by a unique number. This number n is used to specify the histogram type in the output mode '-Hn' under &-PR-OUTPut. Each histogram is described by a number of classes n_c and the center values of the lowest and the highest class, c_l and c_h.

Therefore, the class width is given by

$$\Delta c = \frac{c_h - c_l}{n_c - 1}$$

and the range covered by the histogram is

$$c_l - \frac{1}{2}\Delta c \quad \ldots \quad c_h + \frac{1}{2}\Delta c$$

In paragraph Histograms on page 40 it is explained how COMIS histograms are established.

The units for lowest and highest class depend on the elements which will be output using the histogram type in question. So the lowest and highest class must be given in the actual output User Units. If User Units have been declared before this keyword section, the ranges have be to input in these units. Histogram Types can be used for more than one output variable that can have individual conversion factors: in the output example on page 55 H2 is used for PZ but also for TZ. The conversion factors are listed in the histogram summary.

In the example "Factory" on page 133 the histogram summary is explained in more detail.

NOTE If concentrations are used in the histogram, then this section must come after the section &-POL-DESCription !

Description:

Parameters	Description	Default value	Input format, range
Histogram	Unique histogram number	1	Integer between 1 .. 20
Number of Classes	Number of Classes	100	Integer
Lowest Class	Center of lowest Class	0.0	Real
Highest Class	Center of highest Class	100.0	Real

4.2 Network Description

&-CR (Crack)

```
&-CR       CRACK
  ___________________________________
 |1.| Prefix and Name | Description |
 |__|        (-)      | [-]         |
 |____________________|_____________|

  ____________________________________________________
 |2.|   Cs    |Exp n   | Length  |  Wall Properties  |
 |__|         |        |         |Thickness| U-value |
 | (airl)     |  (-)   |   [m]   |   [m]   | [W/m2/K]|
 |____________|________|_________|_________|_________|

  ____________________________________________________________
 |3.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
 |__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
 |_____________|__________|__________|__________|__________|
```

Example

```
*CR1    Window_cracks
 0.01    0.667
 0.0
*CR2    Door_cracks
 0.03    0.60
 0.0
*CR3    Wall_crack
 0.005   0.85
 0.0     0.5     0.9
*CR4    Crack Example
 0.003   0.667
 0.0     0.3     0.7
```

The mass flow coefficient Cm used in the simulation is calculated with Cs and the length l of the crack from above (data block 2, column 1 and 3), and the actual value a in `&-NET-LIN`ks, column 8 (see page 100).

$$Cm = \frac{a}{l} \cdot Cs$$

Description:

Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘’	15 character string, starting with *CR
Description	description of component	‘’	string
Cs	Air mass flow coefficient	0.001 kg/s@1 Pa	Real
Exp n	Air flow exponent	0.65	0.5 . . . 1.0
Length	Crack length in wall plane	1.	Real
Wall Properties			
Thickness	Thickness of the wall	0.0	Real
U-Value	Heat transfer coefficient	0.3	Real
Filter 1 . . . Filter 5	Filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

The default value for the length is $l = 1$.

NOTE The length of the crack given here and the actual value for the crack length for a certain link in `&-NET-LIN`ks, column 8 must be chosen so that the desired value for Cm is obtained. If no temperature correction has to be calculated, it is perfectly satisfactory to give a "length" of 1.0 for a crack that is in fact 100 m and an actual value of 1.0, for this leads to

$$Cm = Cs$$

In a building with only slightly different leakages in the various facades or windows, giving values of 1.1, 0.95, 1.3, and so on for the actual value can provide a very realistic simulation while only one (average or normalized) crack has to be defined in the section `&-CR`.

A temperature correction factor can be calculated to take into account the difference of the actual air temperature and the temperature at which the crack parameters were measured (the latter temperature is given in `&-NORMCR`, see page 64). This correction factor ranges from 0.9 to 1.15 additionally depending on the air flow exponent n. With air flow exponent n=0.65 it is almost constant=1. (see [14], section 2.2.1.2.6.4 "Walls") This temperature correction will only be initiated if the last two parameters of the second data block, the thickness and the U-value of the wall are entered. Together with the crack length they are used to determine the actual air temperature in the crack. Therefore, real crack lengths must be given if temperature corrections shall be calculated.

The values given for filters filter1 ... filter 5 represent the filter effect, i.e. the capability to hold back pollutants; it is expressed as the fraction of the pollutant that will not pass through the link. filter1 = 0.3 means 30% of the first pollutant defined in `&-POL-DES`cription disappear in the link, 70% go through.

NOTE At least one filter value must appear for *each* crack.

The most reliable values for air mass flow coefficients and exponents are derived from building airtightness data from blower-door tests. The best general source for air mass flow coefficients and exponents is the AIVC's numerical database. A useful summary of the database is given in Technical Note 44 [1].

&-NORMCR (Standard condition for crack data)

```
&-NORM-CR Standard temperature for crack data
 ------------------------------------------------------------------------
|Standard Temperature| Standard Barometric Pressure | Standard Humidity |
| for the Crack Data |      for the Crack Data      |  for Crack Data   |
|   (default 20 C)   |     (default 101.325 kPa)    | (default 0 g/kg)  |
|     [temp]         |           [kPa]              |      [humi]       |
|____________________|______________________________|___________________|
```

Example

```
18      101.320     5
```

This data section is used only for the optional crack temperature correction calculation (see keyword `&-CR` on page 62).

The conditions must either reflect the situation during the measurement of the provided crack data or the conditions to which the data has been normalized.

Compared to the default temperature, a lower normalized value will provide lower mass flow rates in the network.

Description:

Parameters	Description	Default value	Input format, range
Standard Temperature for the Crack Data	Standard Temperature for the Crack Data	20°C	Real
Standard Barometric Pressure for the Crack Data	Standard Barometric Pressure for the Crack Data	101.325 kPa	Real > 0
Standard Humidity for Crack Data	Standard Humidity for Crack Data	0 g/kg	Real > 0

&-FA (Fan)

```
&-FA       FAN
# line1= Fan name....   #line2=flag....   #line3=Pminimum....   #line4=C0
# line5 up to maybe line8=data pairs
# the last line is always the filter line
```

1.	Prefix and Name	Description
	(-)	[-]

2.	# Flag: 1,2 = Polynomial C0,..C5, input provided by user
	3 = Interpolation calculated by COMIS
	from data pairs

Flag	Exp Polynom.	RhoI	NfI	Cs	Exp n
(-)	(-)	(kg/m3)	[rpm]	[airl]	[-]

3.	Pmin	Pmax	Slope	Intercept
	(pres)	(pres)	(cflo/pres)	(cflo)

4.	C0	C1	C2	C3	C4	C5
	(cflo)	[cflo/ pres]	[cflo / pres^2]	[cflo/ pres^3]	[cflo/ pres^4]	[cflo/ pres^5]

5.	Fan Curve: Pressure Rise vs. FlowRate, maximum 4 Lines Data Pairs: minimum 3 Pairs, maximum 12 Pairs					
	(pres)	(cflo)	(pres)	(cflo)	(pres)	(cflo)

9.	Filter 1	Filter 2	Filter 3	Filter 4	Filter 5
	(-)	[-]	[-]	[-]	[-]

Example

```
*FA1    Bathroom Exhaust Fan (Data pairs)
   3        2          1.1     1400      0.1           0.6
       d       d          d          d
       d
      10       40             20        35          30       30
      40       20             50        10
      0.0
```

or

```
*FA-EX3 (Coefficients are given to describe fan-curve)
   1        3          1.2      1        0.00001        0.5
       0      57.8       -0.73        42
      42     -0.11       -0.01     -0.8E-7
      0.0
```

NOTE Pressure difference in fan data is defined in opposite direction than in all other air flow components:

Fan:	$\Delta p = p_{\mathrm{To}} - p_{\mathrm{From}}$
other Airflow components:	$\Delta p = p_{\mathrm{From}} - p_{\mathrm{To}}$

In consequence, the fan "blows" in direction of positive flow.

To determine the fan characteristic, COMIS uses either polynomial coefficients input in data block 4 (Flag 1 and 2, for historical reasons, ☺) or interpolates data pairs given under data block 5 (Flag 3). For the capabibility of an interface to supply polynomial coefficients, please refer to the respective user guides!

If the flag is set to 1 or 2, COMIS uses data from column 2 under data block 2, from data block 3 and from data block 4. If the order of the polynomial is n, then coefficients `C0`, through `Cn` must be given. According to the second example, the fan performance is calculated as follows

$$Q = 42 - 0.11\Delta p - 0.01\Delta p^2 - 0.8\,10^{-7}\Delta p^3$$

The validity of the polynomial is limited to the range [`Pmin`, `Pmax`]. Beyond this range, a linear ap-

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘ ’	15 character string, starting with *FA
Description	description of component	‘ ’	string
Flag	Choice of input	1	Integer 1, 2 or 3
Exp Polynom	Order of the polynomial used to describe fan curve; used with flag 1 or 2	5	Real ≤ 5
RhoI	Air density at fan inlet (test condition)	1.2	Real
NfI	Revolution per minute, fan speed used to determine fan curve; see text!	1.0	Real
Cs	Flow coefficient if fan is off	0.005 kg/s@1 Pa	Real
Exp n	Flow exponent if fan is off	0.65	Real
used if flag 1 or 2 is set in column 1 of data block 2			
Pmin	lowest pressure point for polynomial	0.0 Pa	Real
Pmax	highest pressure point for polynomial	200.0 Pa	Real
Slope	slope of the linear extrapolation for polynomial	-0.0005 m^3/s/Pa	Real
Intercept	intercept between the linear extrapolation and the flow axis	0.1 m^3/s/Pa	Real
used if flag 1 or 2 is set in column 1 of data block 2			
C0 ..C5	Coefficients for polynomial	0.0 m^3/s, 0.0 m^3/s/Pa, 0.0 m^3/s/Pa^2, 0.0 m^3/s/Pa^3, 0.0 m^3/s/Pa^4, 0.0 m^3/s/Pa^5	Real
used if flag 3 is set in column 1 of data block 2; may be omitted with flag 1 or 2			
Pressure	Pressure Value	0.0 Pa	Real
Flow rate	Flow rate Value	0.0 m^3/s	Real
last line in data set			
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

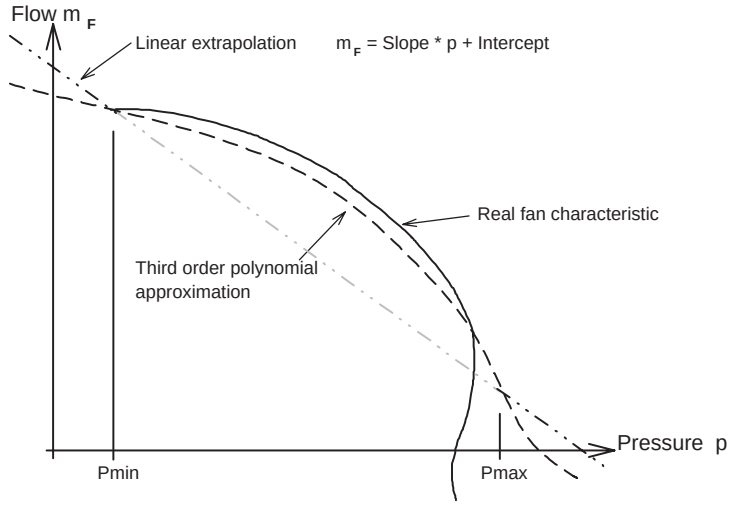


Figure 4.1: Fan characteristic

proximation is used to ensure that the network solver will find a valid solution and not become stuck outside the correctly approximated curve. The slope and intercept have to be calculated outside of COMIS. They are determined by the demand that the polynomial and the linear approximation have equal flows at `Pmin` and `Pmax`, respectively (see figure 4.1).

The pressure range [`Pmin`, `Pmax`] will normally run from 0 Pa up to the pressure where the flow through the fan is zero, as seen from the polynomial. However if the fan characteristic is known for reversed flow (which is unusual for fans) or in the quadrant of reversed pressure (another fan in series over-blows this fan so that it forms a resistance rather than a pump) then the polynomial can be used for the whole range.

NOTE As polynomial fan approximations tend to oscillate, it is recommended to check the polynomial coefficients (Flag 1 and 2) very carefully (with a graph of the polynomial), or to use data pair input (Flag 3) .

If the flag in data block 2 is set to '3', COMIS expects data pairs in line 5 up to the next to last line of a data set. Between the data pairs, a spline interpolation is used to suppress oscillating. The Catmull-Rom spline [5] used here has the property that its curve will pass through the control points it uses. Further, for a point P_i on the curve, the tangent vector at P_i is parallel to the line connecting points P_{i-1} and P_{i+1}. Data lines for data blocks 3 and 4 must be given, but the data are not used (see first example).

The fan speed `NfI` for which the given characteristic is valid, can be input as absolute value in rpm. In this case also the actual fan speed (`&-NET-LINks` section column 8) has to be input in rpm. The other possibility is to normalize the fan speed and set `NfI`=1, then under `&-NET-LINks` the ratio of actual and test fan speed has to be input.

If the fan is off (actual fan speed is less than 1% of the test fan speed `NfI`), the fan will be simulated with a bypass aperture of `Cs` and `Exp n` given in `FA` data block 2 column 5 and 6.

NOTE It should be noticed that in all input data of air flow components the flow is defined as a function of the pressure difference. This is opposite to the definition in data sheets for fans, where pressure difference is a function of the flow. As a consequence it is impossible to input fan curves with a local minimum of $p = f(\text{flow})$ (contraflexure, right part of "Real fan characteristic" in figure 4.1).

NOTE Care has to be taken if data pair input (Flag 3) is used: a missing filter data block is not recognized because the last line is always interpreted as filter data. Subsequently, no error message can be issued.

&-DS (Straight Duct)

```
&-DS          duct straight
# a duct fitting given here will result in an extra dynamic pressure
# loss, Zeta
  ___________________________________
 |1.| Prefix and Name | Description |
 |__|        (-)      | [-]         |
 |____________________|_____________|

 ____________________________________________________________________________________
 |2.|  Ducts straight part                                |       one Fitting    |        |
 |__|_____________________________________________________|______________________|________|
 |  Diam1  |  Diam2  |  Rough  |  Lduct  |  Zeta1  | Type |Param1 |Param2 | Zeta2  |
 |   (m)   |   (m)   |   (mm)  |   (m)   |   [-]   | [-]  | [?]   | [?]   |  [-]   |
 |_________|_________|_________|_________|_________|______|_______|_______|________|

 ____________________________________________________________
 |3.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
 |__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
 |_____________|__________|__________|__________|__________|
```

Example

```
*DS-1     Round duct /w diffusor
   0.3      0.0         0.03      12        1.2       7      0.5      30
   0.0
```

or

```
*DS-1     Rectangular duct
   0.3      0.3         0.03      12        1.2
   0.0
```

The duct is characterized by the values of the duct diameters `Diam1` and `Diam2`. If both are non-zero, a duct with a rectangular cross section is assumed. If only `Diam1` is non-zero and `Diam2` is 0.0, a round duct with a diameter of `Diam1` is assumed. The wall roughness is given in mm. The length of the duct, `Lduct`, must be the real length. `Zeta1` and `Zeta2` represent known dynamic pressure loss coefficients for positive and reverse flow direction respectively. They will be multiplied with

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘ ’	10 character string, starting with *DS
Description	description of component	‘ ’	string
Diam1	Width of duct or diameter	0.1	Real
Diam2	Height of duct	0.0	Real
Rough	Roughness of duct (unit is [mm]!)	0.1	Real
Lduct	Length of straight duct	10.0	Real
Zeta1	Dynamic loss coefficient of duct for positive flow direction	0.0	Real 0.0 . . . 1.0
Type	Type of fitting	0	1 . . . 11, 13
Param1, Param2	See figures 4.2	0.0, 0.0	See table on page 70
Zeta2	Dynamic loss coefficient of duct for reverse flow direction	Zeta1	Real 0.0 . . . 1.0
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

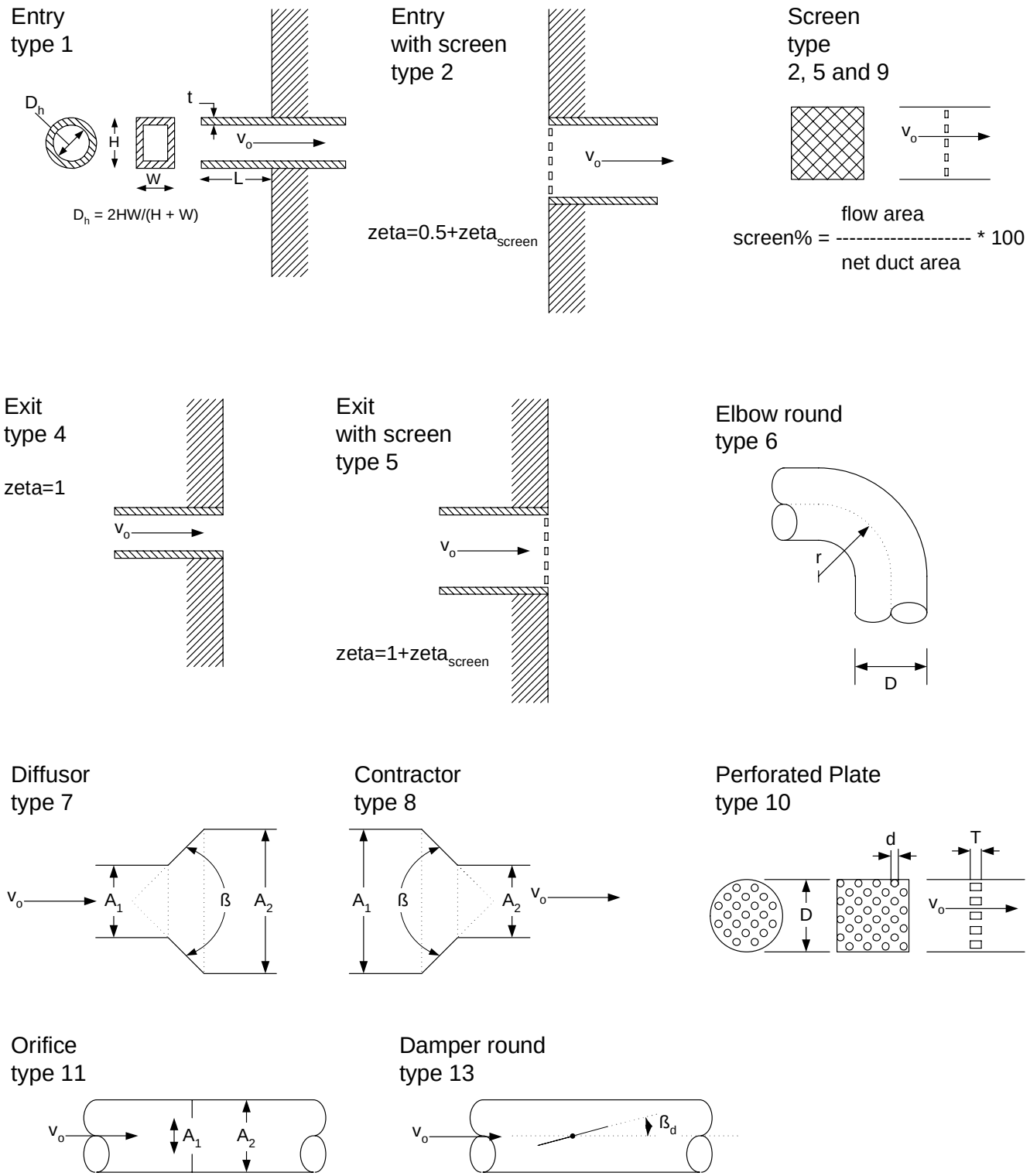


Figure 4.2: Fittings

the 'Actual Value' given under `&-NET-LINks` (see page 100).

Fittings are not available as separate components. However, one fitting can be directly attached

to the duct using the optional parameters `Type`, `Param1`, and `Param2`. The program calculates the dynamic loss coefficient $Zeta_f$ for this fitting according to the given parameters and the actual flow direction. The total pressure loss coefficient of the duct is then:

For flow in positive direction:

$$Zeta = Zeta1 \cdot ActualValue + Zeta_f$$

for flow in reverse direction:

$$Zeta = Zeta2 \cdot ActualValue + Zeta_f$$

Available fittings are shown in figure 4.2. Their parameters are described in the table on page 70.

Type	Name	No of Param	Parameter Description Parameter 1	Parameter 2	Validity, range
1	Entry	2	t/D_h	L/D_h	no limits
2	Entry with screen	1	$screen\%$	-	$screen\% < 100\%$, $L = 0$
3	Hood	2	Type	angle	no limits
	Type 1: round				
	Type 2: rectangular				
4	Exit	0	-	-	-
5	Exit with screen	1	$screen\%$	-	$screen\% < 100\%$
6	Elbow round	1	r/D	-	$0 < r/D < 5$
7	Diffuser round	2	A_1/A_2	β	$A1/A2 < 1$, $\beta < 180^o$
8	Contraction round	2	A_2/A_1	β	$A2/A1 < 1$, $15^o < \beta < 45^o$
9	Obstruction round duct Screen	1	$screen\%$	-	$screen\% < 100\%$
10	Perforated plate	2	T/d	$screen\%/100$	$0.015 < T/d < 0.8$ $0.0 < screen\%/100 < 1.0$
11	Orifice A[a]	1	A_1/A_2	-	$A1/A2 < 1$
12	DIN orifice[b]				
13	Damper round	1	β_d	-	$\beta_d < 75^o$

[a] Routine according to Feustel [12]

[b] This duct fitting is not available

Quantity	Description	Unit
t	Thickness of duct wall	[m]
L	Length of duct inlet	[m]
D	Net diameter of round duct	[m]
D_h	Net hydraulic diameter of duct $D_h = 2HW/(H+W)$	[m]
$H; W$	..Height and Width of rectangular duct	[m]
r	curvature of duct	[m]
A_i	Net area of duct at place i	[m^2]
β	opening angle of duct	[^{o}]
β_d	angle of damper and duct axes	[^{o}]
T	Thickness of perforated plate	[m]
d	Diameter of hole in screen	[m]
$screen\%$	flow area/net duct area*100	[%]

&-F1 (Flow Controller, ideal symmetric)

```
&-F1       FLOWCONTROLLER IDEAL SYMMETRIC
 _____________________________
|1.| Prefix and Name | Description |
|__|        (-)      | [-]         |
|__________________|_____________|

 ____________________________________________________
|2.|          |   Range1                  | Range2       |
|__|          |___________________________|______________|
|     RhoI    |     Cq      |    Exp n    | Fva Setpoint |
|    (kg/m3)  | (cflo@1Pa)  |     (-)     |    (cflo)    |
|_____________|_____________|_____________|______________|

 ___________________________
|3.|   Facade Leakage       |
|__|________________________|
|     Cs      |    Exp n    |
|    (airl)   |     (-)     |
|_____________|_____________|

 ____________________________________________________________
|4.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
|__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
|_____________|__________|__________|__________|__________|
```

Example

```
*F1-Ex1 Example F1 Flowcontroller
     1.2          0.1          0.65          0.2
     0.1          0.65
     0.0
```

For a general description of the flow controller models, please refer to the notes on flow controllers, page 35 ff.

With the component F1 a symmetrical ideal flow controller can be modeled. By “symmetrical” it is meant that the curve will be the same for flows in either direction except for the opposite sign

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘’	10 character string, starting with *F1
Description	description of component	‘’	string
RhoI	Air density at test conditions	1.2	Real
Range 1:			
Cq	Flow coefficient of flow controller (volume flow)	0.1 m^3/s@1 Pa	Real
Exp n	Flow exponent of flow controller	0.65	Real
Range 2:			
Fva Setpoint	Volume flow at setpoint	0.0 m^3/s	Real
Cs	Flow coefficient of facade leakage (mass flow)	0.001 kg/s@1 Pa	Real
ExpN	Flow exponent of facade leakage	0.65	Real
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

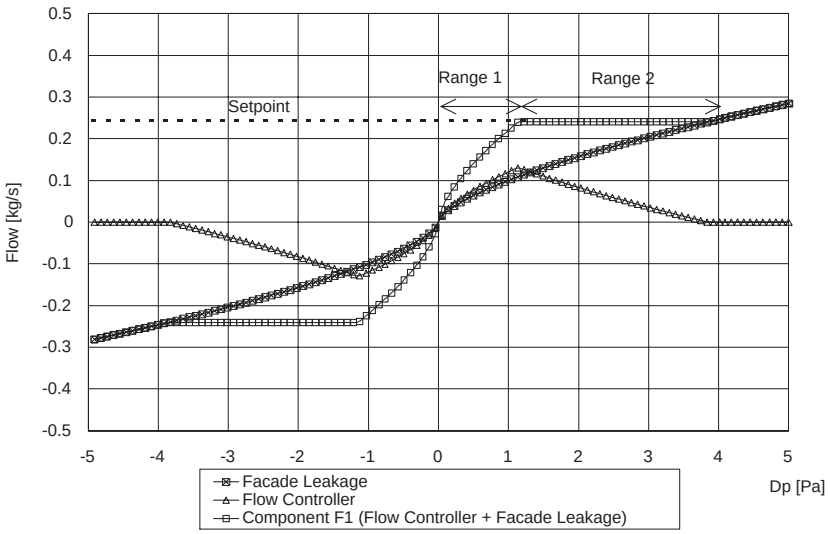


Figure 4.3: Flow controller ideal symmetric (Component F1 with parameters given in the example input)

of pressure and flow. The word "ideal" means that the controller, when used in a facade, has been compensated for the leak of the facade. In reality this means that a blower door test has given values for the air leaks and the flow controller has a chance of adjustment to reduce its flow curve as a function of pressure. The total effect will be that the sum of the flow through the facade and the flow controller is kept constant until the leak flow through the facade exceeds the set-point flow rate. In this case the flow controller is closed and the total flow will increase with increasing pressure according the leakage characteristic.

The component F1 represents the sum of the flow through the facade and the flow controller. That means no additional link for the leakage may be defined in parallel to F1. With this approach the leakage is assumed to be at the same height as the flow controller. In reality this is mostly not true. That's why we have in reality (but not in the model) a small deviation from the ideal constant flow due to different stack pressures.

The resulting flow is temperature compensated according the relation of air densities at actual and test conditions.

Note: The default units of the parameters for range 2 and 3 are volume flow units. With the unit 'cflo' in the `&-PR-UNIT` section they can be changed to mass flow units.

&-F2 (Flow Controller, ideal non-symmetric)

```
&-F2       FLOWCONTROLLER IDEAL NON-SYMMETRIC

 _____________________________
|1.| Prefix and Name | Description |
|__|        (-)      | [-]         |
|__________________|_____________|

 ____________________________________________________________
|2.|          |   Range1                  | Range2 Fva Setpoint |
|__|          |___________________________|_____________________|
|     RhoI    |     Cq      |    Exp n    | positive | negative |
|    (kg/m3)  | (cflo@1Pa)  |     (-)     |  (cflo)  |  (cflo)  |
|_____________|_____________|_____________|__________|__________|

 ___________________________
|3.|   Facade Leakage       |
|__|_________________________|
|     Cs      |    Exp n    |
| (airl)      |     (-)     |
|_____________|_____________|

 ____________________________________________________________
|4.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
|__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
|_____________|__________|__________|__________|__________|
```

Example

```
*F2-Ex1 Example F2 Flowcontroller
     1.2          0.1          0.65          0.2        -0.15
     0.1          0.65
     0.0
```

For a general description of the flow controller models, please refer to the notes on flow controllers, page 35 ff.

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘ ’	10 character string, starting with *F2
Description	description of component	‘ ’	string
RhoI	Air density at test conditions	1.2	Real
Range 1:			
Cq	Flow coefficient of flow controller (volume flow)	0.1 m^3/s@1 Pa	Real
Exp n	Flow exponent of flow controller	0.65	Real
Range 2 Fva Setpoint:			
positive	Positive volume flow at setpoint	0.0 m^3/s	Real
negative	Negative volume flow at setpoint	0.0 m^3/s	Real
Cs	Flow coefficient of facade leakage (mass flow)	0.001 kg/s@1 Pa	Real
ExpN	Flow exponent of facade leakage	0.65	Real
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

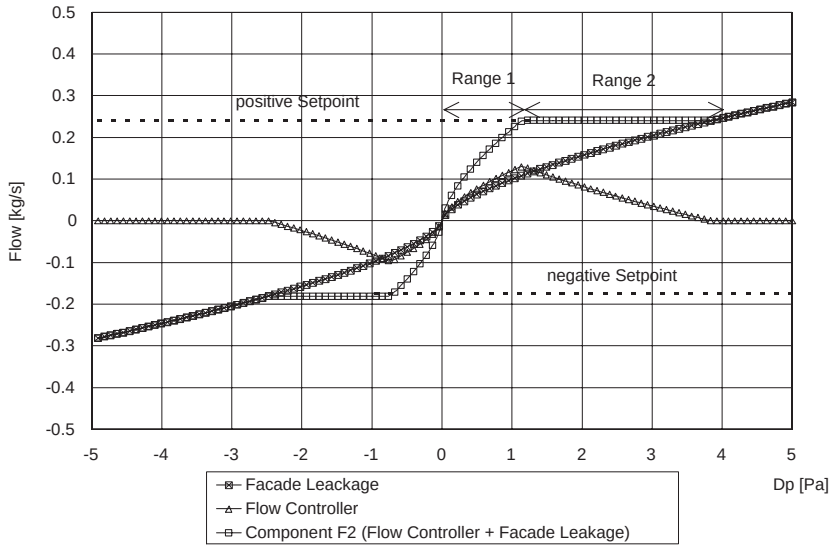


Figure 4.4: Flow controller ideal non-symmetric (Component F2 with parameters given in the example input)

F2 is an asymmetric ideal flow controller which is almost identical to F1 but which has a separate flow rate set-point for negative flow-directions.

Please pay attention to the flow direction when using an asymmetric flow controller in `&-NET-LINKs`. It makes a difference in the result, if the `From` and `To` zones are exchanged. For flow direction and sign conventions, see page 32.

The resulting flow is temperature compensated according the relation of air densities at actual and test conditions.

Note: The default units of the parameters for range 2 and 3 are volume flow units. With the unit 'cflo' in the `&-PR-UNIT` section they can be changed to mass flow units.

&-F3 (Flow Controller, non-ideal symmetric)

```
&-F3       FLOWCONTROLLER NON-IDEAL SYMMETRIC
 _____________________________
|1.| Prefix and Name | Description |
|__|        (-)      | [-]         |
|__________________|_____________|

 ____________________________________
|2.|          |   Range1                |
|__|          |_________________________|
|     RhoI    |     Cq      |    Exp n    |
|    (kg/m3)  | (cfl@1Pa)   |     (-)     |
|_____________|_____________|_____________|

 ____________________________________________________________________________
|3.|  Range2                                                                  |
|__|__________________________________________________________________________|
| R2Dp1  | R2C0   | R2C1    |  R2C2     |  R2C3     |  R2C4     |  R2C5     |
| (pres) | (cflo) | (cflo/  | (cflo/    | (cflo/    | (cflo/    | (cflo/    |
|        |        |   pres) |   pres^2) |   pres^3) |   pres^4) |   pres^5) |
|________|________|_________|___________|___________|___________|___________|

 ____________________________________________________________________________
|4.|  Range3                                                                  |
|__|__________________________________________________________________________|
| R3Dp1  | R3C0   |  R3C1  |  R3C2    |  R3C3    |  R3C4    |  R3C5    | Pmax |
| (pres) | (cflo) | (cflo/ | (cflo/   | (cflo/   | (cflo/   | (cflo/   |(pres)|
|        |        |   pres)|   pres^2)|   pres^3)|   pres^4)|   pres^5)|      |
|________|________|________|__________|__________|__________|__________|______|

 ____________________________________________________
|5.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
|__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
|_____________|__________|__________|__________|__________|
```

Example

```
*F3-Ex1 Example F3 Flowcontroller
     1.2           4.0           0.63
     4.2   5.36   1.41       -7.78e-2    0.0       0.0       0.0
    10.5   0.78   4.41       -0.32       0.0       0.0       0.0       14.0
     0.0
```

For a general description of the flow controller models, please refer to the notes on flow controllers, page 35 ff.

F3 is a symmetric non-ideal flow controller. Non-ideal means that no compensation for other facade leaks is included in the flow controller model. Only the actual characteristic of the controller is considered. The ranges 2 and 3 have to be input with the coefficients of their polynomial approximation.

To evaluate the exact pressures at the intersections of the different curves, the program needs a rough estimation of these values, because the curves could in fact have several points of intersection.

The resulting flow is temperature compensated according the relation of air densities at actual and test conditions.

Note: The default units of the parameters for range 2 and 3 are volume flow units. With the unit 'cflo' in the `&-PR-UNIT` section they can be changed to mass flow units.

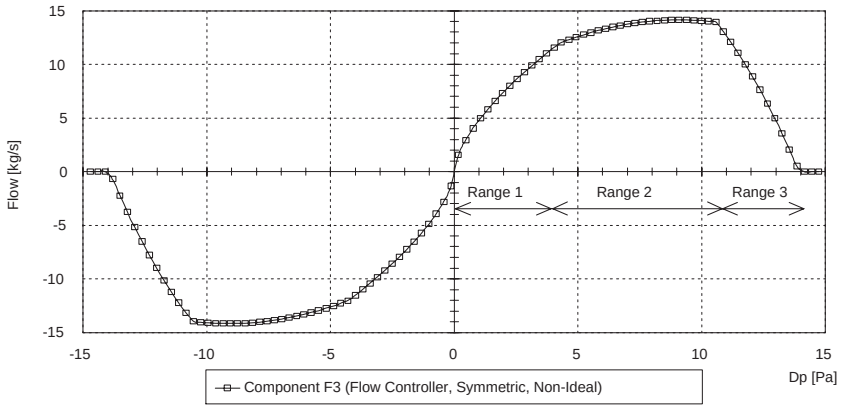


Figure 4.5: Flow controller non-ideal symmetric (Component F3 with parameters given in the example input with decreasing range 3)

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘ ’	10 character string, starting with *F3
Description	description of component	‘ ’	string
RhoI	Air density at test conditions	1.2	Real
Range 1:			
Cq	Flow coefficient of flow controller (volume flow)	0.1 m^3/s@1 Pa	Real
Exp n	Flow exponent of flow controller	0.65	Real 0.5 ... 1.0
Range 2:			
R2Dp1	Estimation for pressure where range 2 starts	0.0 Pa	Real
R2C0 ... R2C5	Polynomial coefficients for range 2 (volume flow)	0.0 m^3/s, 0.0 m^3/s/Pa, etc	Real
Range 3:			
R3Dp1	Estimation for pressure where range 3 starts	0.0 Pa	Real
R3C0 ... R3C5	Polynomial coefficients for range 3 (volume flow)	0.0 m^3/s, 0.0 m^3/s/Pa, etc	Real
Pmax	Maximum pressure for polynomial approximation in range 3. If range 3 is decreasing, above this pressure the flow is zero. If range 3 is increasing, it is proportional to the pressure.	0.0 Pa	Real
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

&-F4 (Flow Controller, non-ideal non-symmetric)

```
&-F4       FLOWCONTROLLER NON-IDEAL ASYMMETRIC
```

1.	Prefix and Name (-)	Description [-]

2.		Range1	
	RhoI (kg/m3)	Cq (cflo@1Pa)	Exp n (-)

3.	Range2 positive flow						
	R2Dp1 (pres)	R2C0 (cflo)	R2C1 (cflo/ pres)	R2C2 (cflo/ pres^2)	R2C3 (cflo/ pres^3)	R2C4 (cflo/ pres^4)	R2C5 (cflo/ pres^5)

4.	Range3 positive flow							
	R3Dp1 (pres)	R3C0 (cflo)	R3C1 (cflo/ pres)	R3C2 (cflo/ pres^2)	R3C3 (cflo/ pres^3)	R3C4 (cflo/ pres^4)	R3C5 (cflo/ pres^5)	Pmax (pres)

5.	Range2 negative flow						
	R2Dp1 (pres)	R2C0 (cflo)	R2C1 (cflo/ pres)	R2C2 (cflo/ pres^2)	R2C3 (cflo/ pres^3)	R2C4 (cflo/ pres^4)	R2C5 (cflo/ pres^5)

6.	Range3 negative flow							
	R3Dp1 (pres)	R3C0 (cflo)	R3C1 (cflo/ pres)	R3C2 (cflo/ pres^2)	R3C3 (cflo/ pres^3)	R3C4 (cflo/ pres^4)	R3C5 (cflo/ pres^5)	Pmax (pres)

7.	Filter 1 (-)	Filter 2 [-]	Filter 3 [-]	Filter 4 [-]	Filter 5 [-]

Example

```
*F4-Ex1 Example F4 Flowcontroller
     1.2           4.0           0.63
     4.2    5.36   1.41       -7.78e-2    0.0       0.0       0.0
    10.5  111.   -20.0         1.0        0.0       0.0       0.0       13.0
    -3.5  -51.4   -0.078      -0.008      0.0005    0.00001   0
   -11.0 -338.   -63.         -3.         0.0       0.0       0.0      -13.0
     0.0
```

For a general description of the flow controller models, please refer to the notes on flow controllers,

page 35 ff.

F4 is an asymmetric non-ideal flow controller. Parameters for range 2 and 3 have to be defined for positive and negative flow directions respectively.

Please pay attention to the flow direction when using an asymmetric flow controller in `&-NET-LINKs`. It makes a difference in the result, if the `From` and `To` zones are exchanged. For flow direction and sign conventions, see page 32.

To evaluate the exact pressures at the intersections of the different curves, the program needs a rough estimation of these values, because the curves could in fact have several points of intersection.

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘ ’	10 character string, starting with *F4
Description	description of component	‘ ’	string
RhoI	Air density at test conditions	1.2	Real
Range 1:			
Cq	Flow coefficient of flow controller (volume flow)	0.1 m^3/s@1 Pa	Real
Exp n	Flow exponent of flow controller	0.65	Real. 0.5 ... 1.0
Range 2 positive flow:			
R2Dp1 (pos.)	Estimation for pressure where range 2 starts	0.0 Pa	Real
R2C0 ... R2C5	Polynomial coefficients for range 2 (positive volume flow)	0.0 m^3/s, 0.0 m^3/s/Pa, etc	Real
Range 3 positive flow:			
R3Dp1 (pos.)	Estimation for pressure where range 3 starts	0.0 Pa	Real
R3C0 ... R3C5	Polynomial coefficients for range 3 (positive volume flow)	0.0 m^3/s, 0.0 m^3/s/Pa, etc	Real
Pmax	Maximum pressure for polynomial approximation in range 3. Above this pressure the flow is proportional to the pressure or zero	0.0 Pa	Real
Range 2 negative flow:			
R2Dp1 (neg.)	Estimation for pressure where range 2 starts	0.0 Pa	Real
R2C0 ... R2C5	Polynomial coefficients for range 2 (negative volume flow)	0.0 m^3/s, 0.0 m^3/s/Pa, etc	Real
Range 3 negative flow:			
R3Dp1 (neg.)	Estimation for pressure where range 3 starts	0.0 Pa	Real
R3C0 ... R3C5	Polynomial coefficients for range 3 (negative volume flow)	0.0 m^3/s, 0.0 m^3/s/Pa, etc	Real
Pmax	Minimum pressure for polynomial approximation in range 3. Below this pressure the flow is proportional to the pressure or zero	0.0 Pa	Real
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

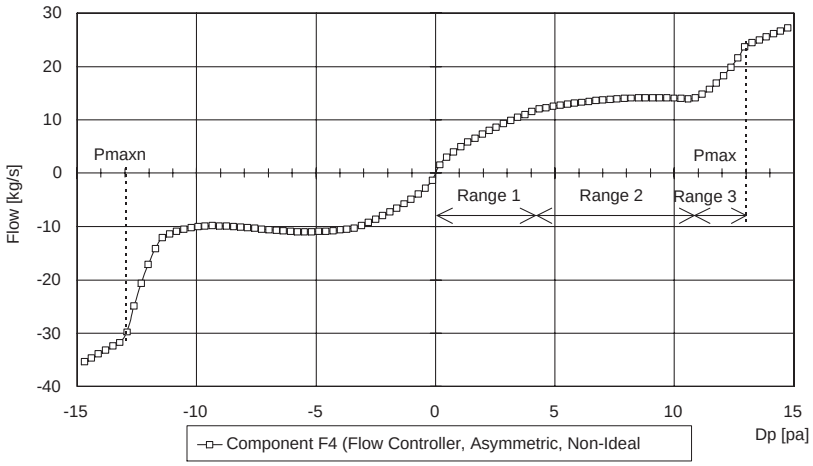


Figure 4.6: Flow controller non-ideal non-symmetric (Component F4 with parameters given in the example input with increasing range 3)

The resulting flow is temperature compensated according the relation of air densities at actual and test conditions.

Note: The default units of the parameters for range 2 and 3 are volume flow units. With the unit 'cflo' in the `&-PR-UNIT` section they can be changed to mass flow units.

&-WI (Windows and Doors)

```
&-WI       Window / Door

  ---------------------------------
 |1.| Prefix and Name | Description |
 |__|        (-)      | [-]         |
 |____________________|_____________|

 ------------------------------------------------------------------
 |2.| Closed:   | Expn | LVO Type     | Lwmax  | Lhmax  |Type specific|
 |__|  Cs       |      |              |        |        |   Length    |
 | (airl/m)     | (-)  | (-)          |  [m]   |  [m]   |    [m]      |
 |______________|______|______________|________|________|_____________|

 ------------------------------------------------
 |3.| Open. | Cd  | Width  | Height | Start Height|
 |__| Fact. |     | Factor | Factor | Factor      |
 |    (-)   | [-] |   [-]  |  [-]   |  [-]        |
 |__________|_____|________|________|_____________|

 ------------------------------------------------------------
 |4.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
 |__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
 |_____________|__________|__________|__________|__________|
```

Example

```
*WI1    Window Example
      0.002        0.57         1        2.2      1.2     1.2
      0.0        0.6    0.0     1.0      0.0
      1.0        0.6    1.0     1.0      0.0
      0.0
```

The air flow component `&-WI` allows to model closed or (partly) opened windows or doors. Five types of "large vertical openings" (LVO) are available:

- ◇ LVO type 1 with rectangular opening with one or more vertical axes,
- ◇ LVO type 2 and 3 with rectangular openings and horizontal pivoting axis,
- ◇ LVO type 4 and 5 with triangular opening area.

Closed window/door

A closed window or door can be simulated either with a simple crack `&-CR` component or with the window air flow component `&-WI`. The difference between these two possibillities is:

- ◇ The crack defined under `&-CR` is located at the link height given in `&-NET-LIN`ks.
- ◇ The crack of a closed window component is distributed over the height of the window ($Lhmax$). The crack length is calculated from the height and the width of the window. LVOs of type 1 that consist of more than one movable part might have an extra crack length which can be defined under $Type\ specific\ Length$.

For a rectangular window/door (LVO type 1 to 3) the crack is distributed as follows:

- ◇ a bottom crack located at link height (crack length $Lwmax$) ,
- ◇ a top crack located at $Lhmax$ above the link height (crack length $Lwmax$) ,
- ◇ a vertical crack uniformly distributed between the bottom and top of the window component, (crack length: LVO type 1: $2 \cdot Lhmax + Type\ specific\ Length$) LVO type 2 and 3: $2 \cdot Lhmax$

For the triangular LVOs type 4 and 5 the crack is distributed as follows:

- ◇ a vertical crack uniformly distributed between the bottom and top of the window component, (crack length: $Type\ specific\ Length$)

The length parameters are defined in figures 4.7 through 4.9. Link heights are described under keyword &-NET-LINks on page 100.

Due to the distribution over the window height it is possible to have a two way flow through the cracks of a closed window.

Open window/door

The &-WI component allows not only the definition of closed windows and doors. In addition, an open window or door can be defined. The state of the component (open, partly open or closed) may vary during a simulation run by defining a schedule for the window (see keyword &-SCH-WIN

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘ ’	10 character string, starting with *WI
Description	description of component	‘ ’	string
Closed: Cs	Flow coefficient for closed windows per m crack length	0.0001 kg/s@1 Pa/m	Real
Exp n	Flow exponent	0.7	Real 0.5 . . . 1.0
LVO Type	Type of Large Vertical Opening 1 = Rectangular LVO 2 & 3 = Horizontal Pivoting Axis LVO 4 & 5 = Triangular LVO	1	Integer 1 ... 5
Lwmax	Width of Window	1.0	Real
Lhmax	Height of Window	1.0	Real
Type specific Lenght	Depending on LVO type, this parameter defines:		
	Type 1: Extra crack length for LVOs with multiple openable parts	0.0	Real
	Type 2 and 3: Height of the pivoting axis (measured from the bottom of the window frame)	0.0	Real
	Type 4 and 5: Total crack length (closed LVO) uniformly distributed over Lhmax	1.0	Real
Opening Factor	Opening factor of window; for LVO Type 2 and 3: Opening Factor corresponds to the opening angle between 0^o and 90^o. Minimum two lines; this table is used for interpolation; the actual value has to be given in &-NET-LINks!	0.0	Real 0.0 . . . 1.0
Cd	Discharge coefficient	0.0	Real 0.0 . . . 1.5
used for LVO type 1, 4 and 5 ignored for LVO type 2 and 3			
Width Factor	Actual Lw/Lwmax	1.0	Real 0.0 . . . 1.0
Height Factor	Actual Lh/Lhmax	1.0	Real 0.0 . . . 1.0
Start Height Factor	Actual Start Height/Lhmax	0.0	Real 0.0 . . . 1.0
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

on page 104).

The geometry of the opening area for the different LVO types is defined as follows:

- ◇ LVO type 1 with rectangular opening area (see figure 4.7),
- ◇ LVO type 2 with a horizontal axis, leading to opening areas as sketched in figure 4.9, upper left part,
- ◇ LVO type 3 with a horizontal axis, with an alternative geometry of the opening areas as sketched in figure 4.9 upper right part,
- ◇ LVO type 4 with triangular opening area vertex at the bottom,
- ◇ LVO type 5 with triangular opening area vertex at the top.

The triangular openings of LVO type 4 and 5 can be used together with the rectangular LVO type 1 to assemble any geometry of opening area by hand. This is often necessary if a window with horizontal axis is positioned closely to a ceiling or a wall or if the window has a reveal. In these cases the opening areas will be different to those defined in LVO type 2 and 3. In cases where the overlapping of the window frames have to be taken into account the assembling by hand might also lead to a more correct opening geometry. Also for open skylights the opening areas have to be assembled by hand as both LVO types 2 and 3 are only allowed to use in vertical position and not in a slanted position as shown for the LVO type 1 in figure 4.8.

The difference between LVO type 2 and 3 for windows with horizontal axis is besides the different geometry of the opening area an improved discretization of the opening area in LVO type 3 which prevents from a discretization error especially for small opening angles, where the top and bottom rectangles tend to totally disappear in LVO type 2. Additionally the program can determine the discharge coefficient Cd for the LVO type 2 depending on the opening angle and the height to width ratio.

Figure 4.9 shows an example of the flow through a window with horizontal axis depending on the opening angle. The flow is calculated with LVO type 2 and 3 and additionally with a CFD model. Although the geometry of the opening areas according to LVO type 3 seems to be more correct, the result of LVO type 3 shows a better agreement with the result from CFD calculation. The agreement of LVO type 3 is further improved with the optional internal Cd value calculation depending on the opening angle.

If the window is opened, the size of the opening will be determined from $Lwmax$ and $Lhmax$, the opening factor, column 1 in data block 3, and — for LVO type 1, 4 and 5 — the parameters given in columns 3, 4, and 5 in data block 3. Opening factor 0.0 means closed, 1.0 means fully open. For LVO type 2 and 3 this range of the opening factor corresponds to the range of opening angle from 0^o to 90^o. Example: opening factor 0.5 means the opening angle is 45^o. For LVO type 2 and 3 the size of the opening is determined completely by the opening factor (see figure 4.9 upper part).

At least one line with opening factor 0.0 and one line with 1.0 must be supplied. There may be additional lines between the two. All values of the opening factor between 0.0 and 1.0 indicate that the parameters following on that line are for a partly opened window. During the simulation the geometry and parameters of the opening at the actual timestep will be determined by interpolation. The actual value of the opening factor for the window will be read first from the Actual Value in `&-NET-LIN`ks and then from schedule `&-SCH-WIN`dow, if existant.

The temperature drop caused by opening an external window can be calculated in case of single sided ventilation, using a simple thermal model of the room. If this temperature drop in a zone shall be calculated, zone thermal properties have to be input under `NET-ZT` (see page 99).

From column 2 of data block 3 the discharge coefficient Cd corresponding to the opening factor of that line is read. There is an option to calculate Cd internally, see paragraph “Calculation of Discharge Coeffidient” on page 86.

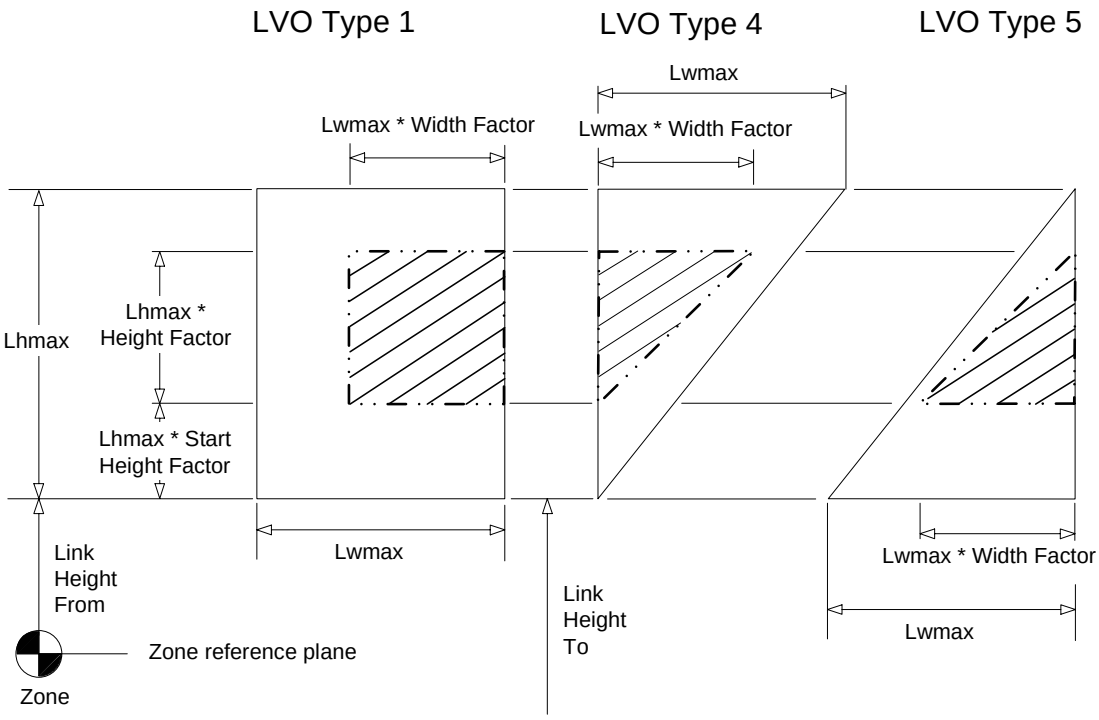


Figure 4.7: Windows and doors, parameters for LVO Type 1, vertical window or door.
For the meaning and input of link heights, see keyword &-NET-LINks on page 100

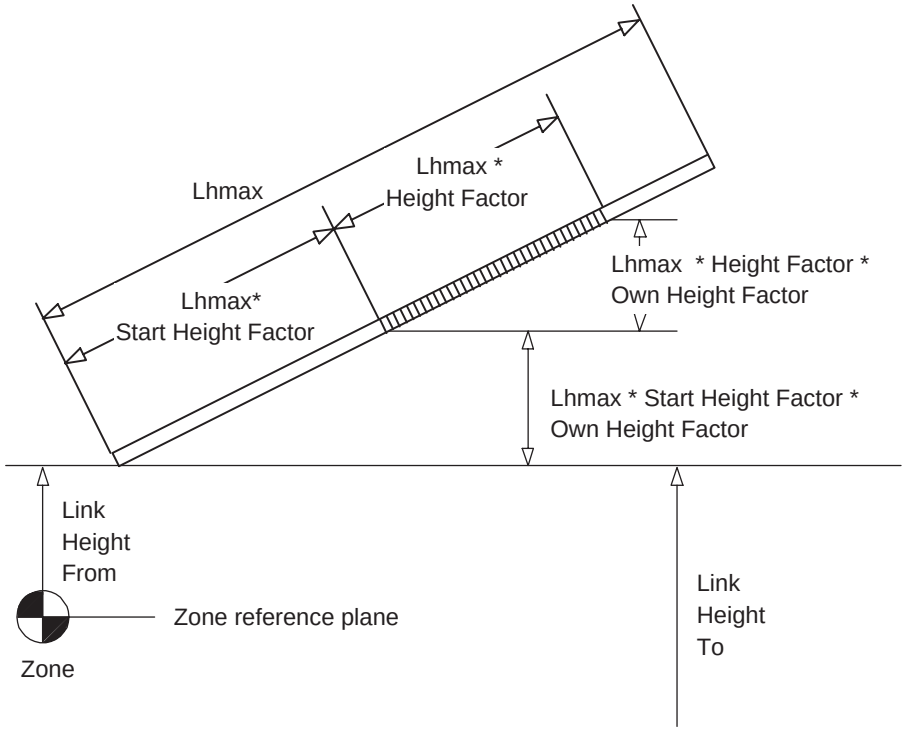


Figure 4.8: Windows and doors, parameters for LVO Type 1, slanted window (or door).
The $OwnHeightFactor$ is input in &-NET-LINks. For the meaning and input of link heights, see keyword &-NET-LINks on page 100

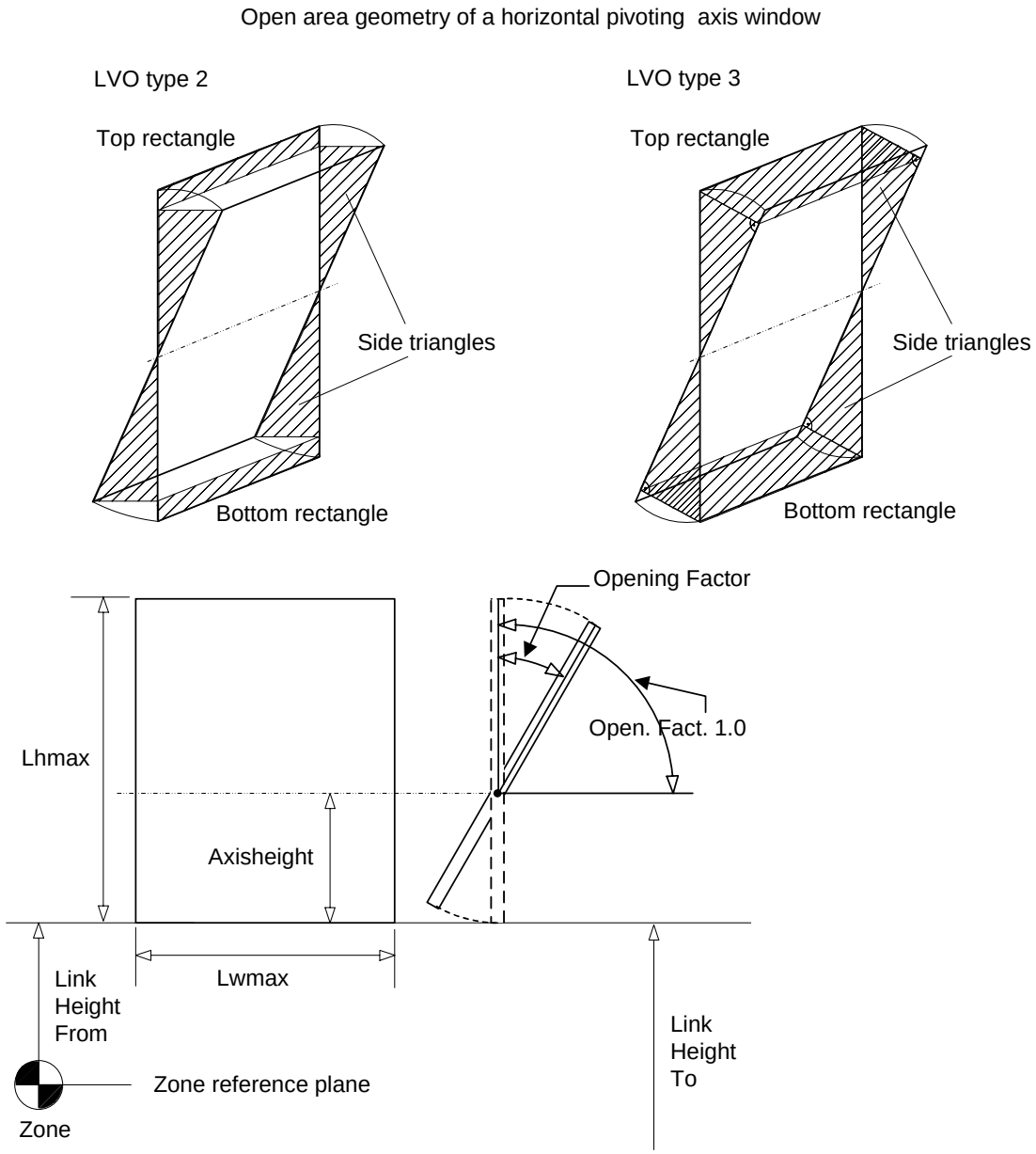


Figure 4.9: Windows and doors, parameters for LVO Type 2, window with horizontal pivoting axis. In the model, the air flows through the hatched area (opening size = top rectangle + bottom rectangle + side triangles).
For the meaning and input of link heights, see keyword `&-NET-LIN`ks on page 100

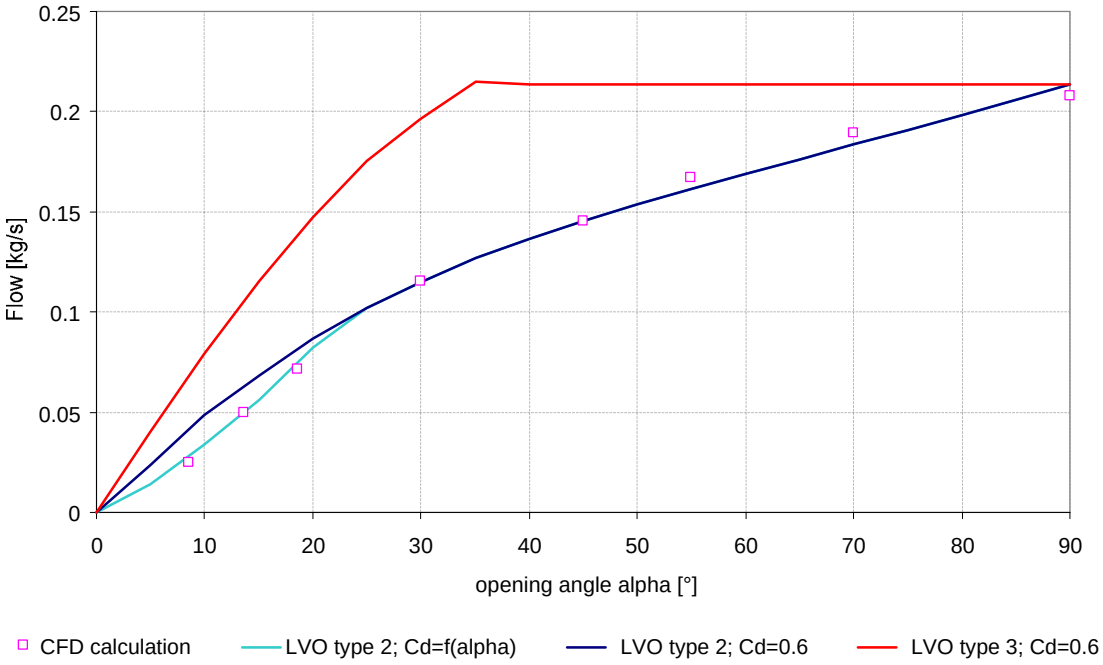


Figure 4.10: Temperature difference driven flow through a window with horizontal pivoting axis depending on the opening angle modelled with LVO type 2 and 3 and with CFD.
(Window: height = 1.14m; width = 0.78m; temperature difference: 24 K)

The geometry of a partly opened horizontal pivoting axis window (LVO type 2 and 3) is determined completely by its opening factor (see figure 4.9). For openings with vertical axes (LVO type 1) and for triangular openings (LVO type 4 and 5) there must be additional factors which define the size and position of the opening relative to $Lwmax$ and $Lhmax$:

Opening width: $= Lwmax \cdot WidthFactor$

Opening height: $= Lhmax \cdot HeightFactor$

Start height: $= Lhmax \cdot StartHeightFactor$

These parameters are input in columns 3, 4 and 5 of data block 3.

Start Height is the distance between the bottom of the window frame and the bottom of the actual opened part of the window (see figure 4.7). The sum of Height Factor and Start Height Factor must be less than or equal to 1.0 in order to have the actual opening within the window frame.

Slanted windows may only be of LVO type 1, 4 and 5. For slanted windows (e.g. windows in a roof) the *Own Height Factor* (see figure 4.8), which is the cosine of the angle between the window plane and a vertical plane, must be given in `&-NET-LIN`ks (see page 100). With this factor the program will calculate the start height and the height of the opening in order to get the correct stack pressure difference profile. For calculating the area of the opening, the *Own Height Factor* will not be taken into account because the flow is assumed to be perpendicular to the opening plane. With this definition it is even possible to input an opening which is horizontal ($Own\ Height\ Factor = 0$).

NOTE Slanted openings of LVO type 2 and 3 ($Own\ Height\ Factor < 1$) are not allowed in the current version. Skylights in a roof have to be modelled by assembling the opening area by hand using rectangular and triangular openings.

NOTE With $Own\ Height\ Factor = 0$, only one-way flow is possible, because the pressure difference is constant throughout the opening. This means that single sided ventilation cannot be modelled for $Own\ Height\ Factor = 0$.

Calculation of the Discharge Coefficient

If the user inputs $Cd = 0$ in column 2 of data block 3, the program will determine the discharge coefficient Cd from the other input parameters:

For internal doors (LVO type 1): Cd is a function of the ratio height of the door/height of the room (see [24]); Cd will be in the range from 0.05 to 0.5.

Measurements conducted by CSTB led to the following model equation:

$$Cd = 0.609 \cdot Hrel - 0.066$$

where $Hrel = Hdoor/Hroom$ and $0.2 < Hrel < 0.9$

For external LVO type 1: Cd is a function of wind velocity, temperature difference, and depth of the room (see [7]); Cd varies between 0.6 and 1.5.

For LVO type 2: Cd is a function of the opening angle α and the height to width ratio and varies in the range of 0.2 to 0.6 (see figure 4.10).

Calibration with results from CFD calculations led to the following equation [6]:

$$Cd = 0.0147 \cdot \alpha - 0.0928 \cdot \frac{Lhmax}{Lwmax} + 0.4116$$

The calculated Cd values can be output with the otput option CL under the &-PR-OUTP keyword (see page 55)

If Cd is determined by the program, the following restrictions apply for the input:

- LVO must be type 1 or 2.
- $Cd = 0$ for all opening factors.
- In addition for internal doors,
  - the doorstep must be lower than 0.05 m,
  - the zone reference height under `&-NET-ZON`es (see page 92) has to be at floor level and must be the same in the `From` and the `To` zone,
  - the zone volume must be input with the `H/D/W` form under `&-NET-ZON`es (see page 92); the height H has to be the same for the `From` and the `To` zone.
- In addition for external openings,
  - the opening must be linked to an external node under `&-NET-LIN`ks (see page 100), not to a specified pressure,
  - the zone volume must be input with the `H/D/W` form under `&-NET-ZON`es (see page 92),
  - only one open window or door per zone is allowed (one with opening factor different from 0).

&-TD (Test Data)

```
&-TD       TEST DATA COMPONENT

 _____________________________
|1.| Prefix and Name | Description |
|__|        (-)      | [-]         |
|_____________________|_____________|

 _______________
|2.|   RhoI      |
|__|  [kg/m3]    |
|_______________|

 ____________________________________________________________
|3.| Pressure and Flowrate, maximum  6 Lines                  |
|__| Data Pairs: maximum 18 Pairs                             |
| (pres)  | (cflo)  | (pres)  | (cflo)  | (pres)  | (cflo)  |
|_________|_________|_________|_________|_________|_________|

 ____________________________________________________________
|4.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
|__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
|_____________|__________|__________|__________|__________|
```

Example
```
*TD-1   Testdata
       1.2
     10.1   1.0          20.0   1.5          30.3   1.9
       0.0
```

The Test Data component is there so that measured results can be used in COMIS without having to define a separate routine for this. Care should be taken if flow is descending with ascending pressure difference. This leads to negative nodes and therefore convergence will be destroyed (see section: “Notes on Convergence” on page 44).

RhoI is the density at which the (volume) flow rates were measured. By default, flow data is expected to be volume flow with [m^3/s]. The flow unit for test data can be changed under `&-PR-UNITS` (see page 49) by giving the desired unit using the keyword `cflo`. By chosing the appropriate unit, mass flow data can be used, too.

If one data pair is input, a constant flow at all pressures is assumed. If two or more pairs are given, a linear interpolation between successive data pairs is made if the actual pressure is within the pressure range spanned by the data pairs. A linear extrapolation is made outside the maximum

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘ ’	10 character string, starting with *TD
Description	description of component	‘ ’	string
RhoI	Density of air at entry point	1.2	Real
Test data pairs (have to be given in ascending order for pressures)			
Pressure	Pressure Value	0.0 Pa	Real
Flow rate	Flow rate Value	0.0 m^3/s	Real
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

and minimum data. It is assumed that the data pairs are given in ascending order for pressures.

The flow rate given here can be scaled with the Actual Value given under `&-NET-LIN`ks (see page 100).

NOTE Be aware that the pressure over a link is the pressure difference `From`-zone minus `To`-zone. This means that a crack has data like

$$(p, q) = (-10, -0.05) \qquad (p, q) = (0, 0) \qquad (p, q) = (10, 0.05)$$

A fan has its curve in the 2nd quadrant, like:

$$(p, q) = (-100, 0) \qquad (p, q) = (-50, 0.05) \qquad (p, q) = (0, 0.07)$$

This fan blows in the `To`-zone and sucks in the `From`-zone.

For flow directions and sign conventions see page 32.

&-RF (Related Flow Component)

```
&-RF       Related Flow COMPONENT

 _________________________________
|1.| Prefix and Name | Description |
|__|        (-)      | [-]         |
|____________________|_____________|

 ___________________________
|2.|Unit Flag   |  RhoI     |
|__| 1: ?=cflo  |           |
|    2: ?=flow  | [kg/m3]   |
|_______________|___________|

 ____________________________________________________________
|3.| Reference Flowrate and related flowrate, max 6 Lines    |
|__| Data Pairs: minimum 3 Pairs, maximum 18 Pairs           |
|   (?)    |  (?)    |  (?)    |  (?)    |  (?)    |  (?)    |
|__________|_________|_________|_________|_________|_________|

 ____________________________________________________
|4.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
|__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
|_____________|__________|__________|__________|__________|
```

Example

```
*RF1    Flowhood
             2         1.2
# 2 means the flows given below are volumetric flowrates
#    pair  1              pair  2               pair  3
     -0.1     0.2          0.02     0.2           0.045     0
#    pair  4
     0.4      0
     0.0
```

The component name RF stands for Related Flow. The flowrate through this component is a function of the flow through a reference link. The link to be used as reference link is given at &-NET-LINKs (see page 100) in the column where schedules and reference links for duct junctions

Description:			
Parameters	Description	Default value	Input format, range
Prefix and Name	unique identifier	‘ ’	10 character string, starting with *RF
Description	description of component	‘ ’	string
Unit Flag	Flag=1 : cflo unit is used; Flag=2 : flow unit is used for both Reference and Related flow rate	0	Integer 1 or 2
RhoI	Density of air at entry point	1.2	Real
Related flow data pairs			
Reference Flow Rate	Reference flow rate	0.0 m^3/s	Real
Related Flow Rate	Component flow rate, function of reference flow rate	0.0 kg/s	Real
Filter 1 . . . Filter 5	filter effect; capability to hold back pollutants 1 . . . 5	0.0	Real 0.0 . . . 1.0

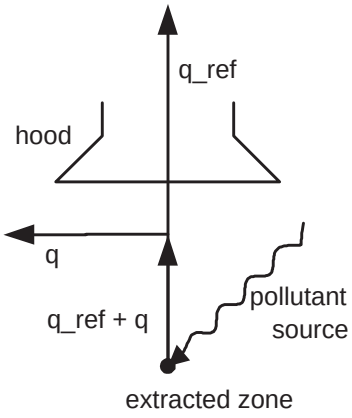


Figure 4.11: Related Flow Component, flows for example kitchen hood.

are defined.

To avoid convergence problems, the Related Flow component has no flow rate derivative. This means that a zone cannot be linked only with RF components, there must also exist some normal links, like a crack or a fan.

The component has been developed to model kitchen hoods. Therefore, the examples given here deal with kitchen hoods. Alternative ways to model such components are shown in figure 4.12.

If a given hood efficiency is known as a function of the hood extract flowrate (through link 3) then the spread flowrate (link 4) can be modelled with the component RF.

For a kitchen hood the reference flow q_{ref} is the flow through the hood, an eventually built in fan and the following duct. The (unwanted) flow from the cooking range that is leaking/circulating out of the hood into the kitchen zone is given by the RF type. If q is the flow leaking to the kitchen zone, the efficiency of the hood η_h can be defined as

$$\eta_h = \frac{q_{ref}}{q_{ref} + q} \cdot 100\%$$

for positive q and positive q_{ref}.

The efficiency for the data pairs presented in the example above range from 9.1% to 100%.

# pair	efficiency	
# 1	–	
# 2	9.1%	(flow into the room is much more than the hood extract flow)
# 3	100%	
# 4	100%	

Between pair 2 and pair 3 is the normal operation range of the hood. The pairs 1 and 4 have been added to make well defined extrapolations.

How a hood is being integrated into the network is shown in the section `&-NET-LIN`ks on pages 103ff. The link to be used as reference link is given at `&-NET-LIN`ks in the same column where schedules and reference links for duct junctions are given.

As shown in the lower part of figure 4.12, it is recommended to use these RF types always in pairs with opposite flow directions to force balanced flow. In this case there is no effect on the flow balance during the iteration process. Nevertheless it will result in the desired flow exchange that can spread pollutants.

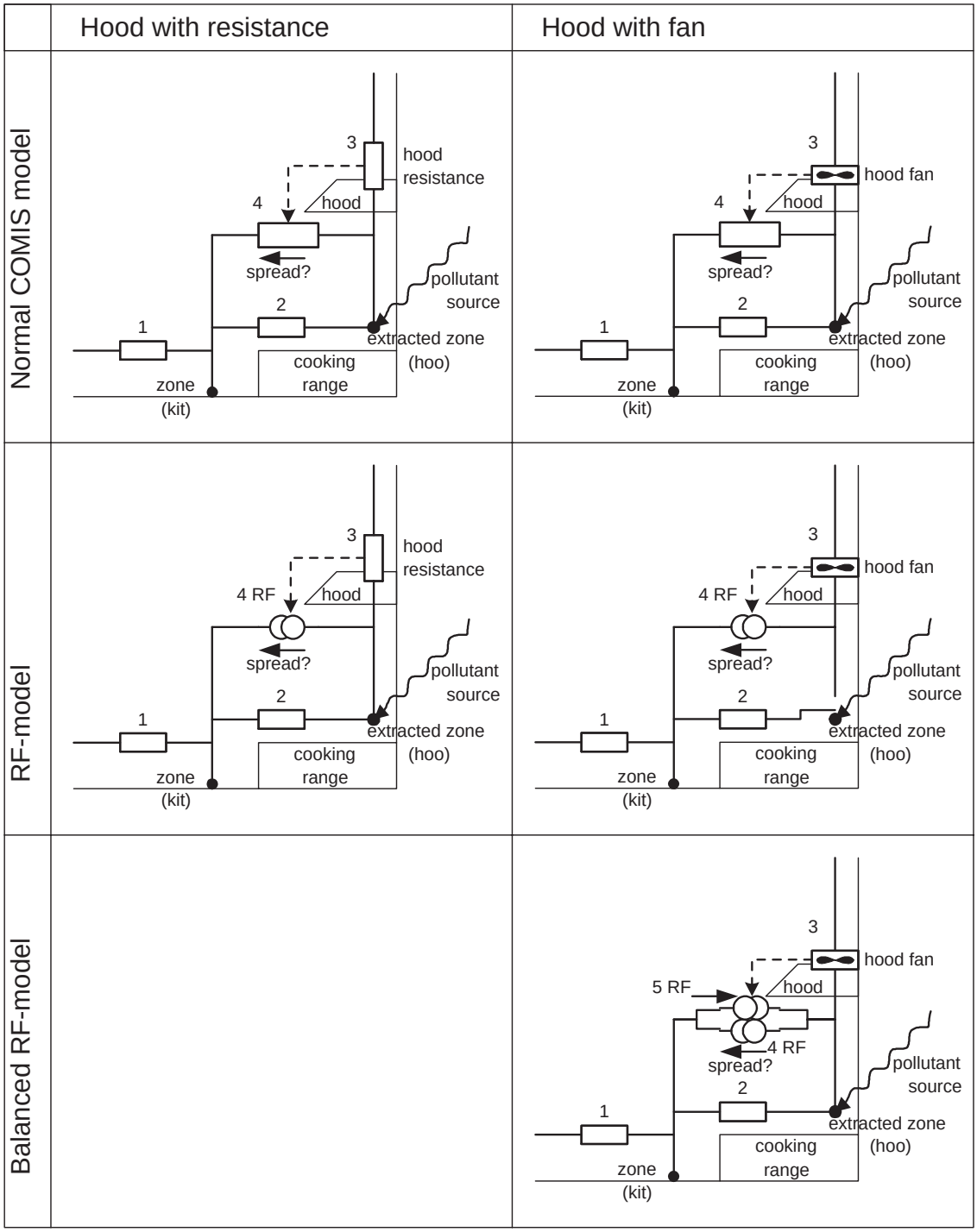


Figure 4.12: Kitchen Hoods
Five ways to simulate a kitchen hood in COMIS are shown in this figure. In the normal COMIS model link 4 is a normal opening (CR) or a large opening. The zone above the cooking range is given a higher temperature, and spread can occur to the kitchen if the flow through the hood (link 3) is insufficient. The same setup is shown to the right except that there is a fan in the hood. In both cases the spread simulated by COMIS, is caused by the stack pressure between link 2 and the higher positioned link 4. The corresponding RF-models are shown beneath the normal models.

&-NET-ZONes (Zones Definition)

```
&-NET-ZONes
 ------------------------------------------------------------------------
|Zone| Name          | Temp |Ref.    |   Volume     | Humid.| Schedule |
| ID |               |      | Height|   [m3]       |       |  Name    |
|    |               |      |       |   H/D/W      |       |          |
| (-)| [-]           |[temp]| [m]   |   [m/m/m]    |[humi] |  [-]     |
|____|_______________|______|_______|______________|_______|__________|
```

Example

```
 E01  Kitchen        18.1    -0.85     100.        10.0     T1/H1
 W02  Living1        20.      0.50     200.        12.0
 N04  Bedr.Child     19       2.85     150          0.0     H3
 N05  Dining         21       0.15    2.4/3.3/4.0   1.5
 a2   auxiliary_node  d        d         0.01
```

The height of the zone reference plane, measured from the building reference plane must be at the floor level of the room

- ◇ if the zone is layered or
- ◇ if temperature drop in a single-sided ventilated zone has to be calculated
- ◇ if it has an opening for which Cd shall be calculated.

For details, please see figure 4.15.

The volume of the zone must be input as Height/Depth/Width

Description:

Parameters	Description	Default value	Input format, range
Zone ID	Zone Identification	‘ ’	String 20 char
Zone Name	Name of specific zone	‘ ’	String 20 char, but no blanks between words
Temp	Temperature setting in zone	20.0^{o}C	Real
Ref. Height	Height of zone reference plane measured from ground plane (see figure 4.15)	0.0	Real
alternative volume input:			
Volume	Volume of zone	75.0	Real
H/D/W	Height/Depth/Width of the room.	3.0/5.0/5.0	Real/Real/Real
Humid.	Humidity in zone; 0 if H2O is used as pollutant.	0.0 g/kg	Real
Schedule Name	Names of schedules applicable to zones. The first character of the name indicates the schedule type: T : Temperature schedule H : Humidity schedule S : Sink schedule Q : Source schedule Use ‘/’ to separate schedules; no blanks or commas allowed	‘ ’	String 50 char

- ◇ if the zone is layered, or
- ◇ if the temperature drop in a single-sided ventilated zone ist to be calculated, or
- ◇ if it has an opening for which Cd shall be calculated, or
- ◇ if fictive sources and their room size dependencies shall be used.

NOTE The humidity in the zone must be input here only if it shall be used for calculating the air density. It must be set to zero in all zones if it is defined as pollutant under `&-POL-DES` and under `&-NET-ZP`, and transport mechanism shall be studied. For details, see page 36.

It is possible to have more than one source and/or sink schedule per zone.

&-NET-ZL (Zone Layers)

```
&-NET-ZL zone-layers
 ------------------------------------------
| Zone  | Start  | Temp.  |  Hum.  | Poll.  |
| *ID   | Height | Grad   | Grad   | Grad   |
|       |        |        | Factor | Factor |
| (-)   |  (m)   |[temp/m]|  [1/m] |  [1/m] |
|_______|________|________|________|________|
```

Example

```
*E01        0.      0.       0.       0.
*1          -.25    2.0      0.10     0.0
            1.5     3.0     -0.15     0.0
*123        1.1     1.5      0.0      0.0
```

A zone may be devided in sub-zones by defining layers. Layers can be defined for any number of zones. There is no limit to the number of layers per zone as long as the the total number of layers does not exceed the maximum of (currently) 50. The layers must be given per zone, in a rising sequence. Every next layer must have a start height which is higher than the one of the previous layer in the same zone. The start height is measured relative to the zone reference plane (positive above zone reference plane).

Layers can start anywhere in the zone including outside the range of the floor and ceiling. The following restrictions apply to input under `&-NET-ZONE`s (see page 92) for layered zones:

- ◇ the zone reference height has to be at floor level, and
- ◇ the zone volume must be input with the `H/D/W` form.

Starting from the values in the zone reference plane, the profiles are established using the gradients within one layer to determine the values at the start height of the next layer. According to figure 4.13, the concentration gradient m of pollutant no 1 in a layer l is the product of the `Poll. Grad Factor` m_l and the concentration c_0 in the zone reference plane. The concentration c_{l+1} at the start height h_{l+1} of the next layer is

$$c_{l+1} = c_l + m \cdot (h_{l+1} - h_l)$$

Description:

Parameters	Description	Default value	Input format, range
Zone ID	Identification of zone in which layers are defined. The zone must be defined!	‘ ’	String
Start Height	Height of the lower boundary of the individual layer measured from the zone reference plane	0.0	Real
Temp. Grad	Temperature gradient within specific layer	0.0 ^{o}C/m	Real
Hum. Grad Factor	The humidity gradient is the product of Hum. Grad Factor and the moisture content in the zone reference plane.	0.0	Real
Poll. Grad Factor	The pollutant gradient m is the product of Poll. Grad Factor m_l and the pollutant concentration c_0 in the zone reference plane. The data refer to the pollutant defined as No 1.	0.0	Real

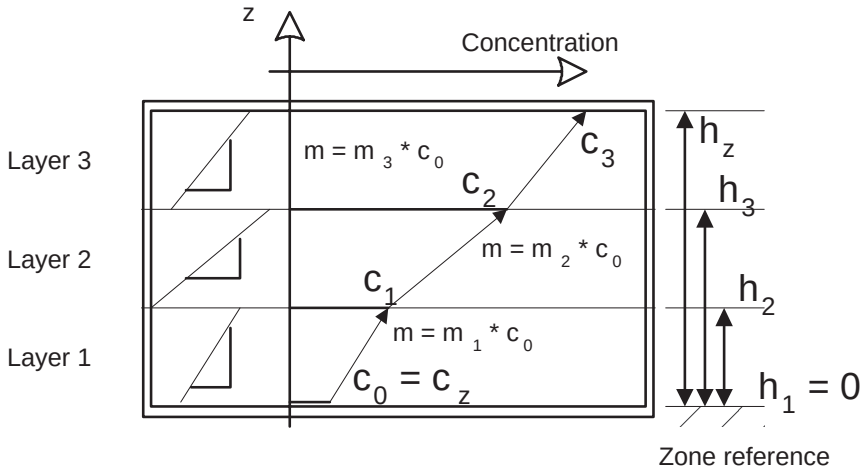


Figure 4.13: Construction of concentration profile in layered zone.

The gradients are used to enable height dependent driving forces (temperature, concentrations). However, they are not influenced by the resulting flows. For the next time step, the zone volume is first assumed to be fully mixed, then the gradient is used to calculate layer characteristics. No further exchange between layers is taken into account.

NOTE Layers may not be defined for single sided ventilated zones.

&-NET-ZP (Zone Pollutants)

```
&-NET-ZP zone-pollutants
```

Zone	Pollutant			
*ID	Initial Concentration	Source	Sink	Decay
(-)	(conc?)	[sour?]	[sink?]	[1/s]

Example

```
*E01        0.              0.          0.          0.

*E02        0.              0.          0.          1.E-4

*001        0.00            0.00        1.005       0.
            0.00            1.25        0.000       0.
            0.00            0.00        1.005       0.
```

The initial values for pollutant concentrations can be given together with the nominal value of the source strength, sink strength and a decay constant. Exponential decline of the pollutant concentration due to e.g. absorption processes can be modelled with the decay constant.

The first data line refers to the pollutant with no. 1 defined in the pollutants definition data section `&-POL-DES`cription (see page 123). The second data line refers to pollutant with no. 2, and so on.

NOTE To run the pollutant module, appropriate input must be given in several data sections:

- ◇ `&-PR-SIMU` (see page 52) keyword `POL [<no>]` to enable pollutant transport for all or the given number of pollutants
- ◇ `&-NET-ZP` (this data section) at least data for one pollutant in one zone must be given here to activate the pollution routines
- ◇ `&-POL-DES` (see page 123) definition of pollutants

Description:

Parameters	Description	Default value	Input format, range
Zone ID	starting with '*', followed by the identification of a defined zone	' '	String
Pollutant			
Initial Concentration	Concentration of specific pollutant at the beginning of the simulation; mass or volume of pollutant per mass of clean, dry air (see page 36)	0.0 kg/kg	Real
Source	Nominal source strength	0.0 kg/s	Real
Sink	Nominal sink strength	0.0 kg/s	Real
Decay	Zone and pollutant specific decay constant of pollutant due to e.g. absorption or adsorption (will be added to the pollutant specific decay constant defined under `&-POL-DES`cription (see page 123))	0.0 1/s	Real ≥ 0.0

Optionally, schedules `&-SCH-SIN` and `&-SCH-SOU` can be defined for pollutants to change sink and / or source strength.

NOTE The humidity must be set to zero in all zones under `&-NET-ZON` if 'H2O' is defined as pollutant and transport mechanism shall be studied. For details, see page 36.

&-NET-EXTernal (External Node Data)

```
&-NET-EXTernal node data
 ---------------------------------------
| External   | Facade     | Outside     |
|  Node Name |  Elem Name | Conc Factor |
|   (-)      |   (-)      |   [-]       |
|____________|____________|_____________|
```

Example

```
1a          2          1.0
1b          2          0
2           5          1
3           7          1
4           9          1.1
```

The building is linked to its ambiance via external nodes, i.e. nodes which are just outside the building. External nodes are associated to facade elements of the building (see Cp value input under &-CP-VALUes and building orientation under &-ENV-BUIlding). Wind pressure data are defined for facade elements.

The distinction between external nodes and facade elements allows for having more than one node per facade element. Therefore, one facade element may have different outside concentration factors for different links.

Outdoor pollutant concentrations can be activated by defining a schedule for them (see &-SCH-POL). The outside concentration factor can modify these global concentration for each node individually.

NOTE The section &-NET-EXTernal must appear before the section &-CP-VALues

Description:

Parameters	Description	Default value	Input format, range
External Node Name	External node name. This name or number appears with a preceding minus sign in the part &-NET-LINks	'1'	String 20 char
Facade Element Name	Name or number of related facade element. Cp values are linked to the respective facade element	'-'	String
Outside Concentration Factor	Ratio of outside pollutant concentration at this external node and outdoor pollutant concentration	1.0	Real

&-NET-ZT (Zone Thermal Properties)

```
&-NET-ZT zone thermal properties

  ------------------------------------------------
 | Zone  | Conduc- | Density | Capacity| Wall    |
 | *ID   | tivity  |         |         |Thickness|
 | (-)   | (W/mK)  | (kg/m3) | (J/kg/K)| (m)     |
 |_______|_________|_________|_________|_________|
```

Example
```
*E01       0.8       1420      1000      0.2
```

Zone thermal properties are required to estimate the temperature drop in a zone caused by opening an external window. Only for single sided ventilation, this temperature drop can be calculated using a simple thermal model of the room. The walls are characterized by their thermal diffusivity. No conduction to adjacent zones is taken into account. The thermal properties are mean values of all walls in the zone.

The thermal model is only valid for openings of LVO type 1 (rectangular vertical opening). If a window with LVO type 2 (horizontal pivoting axis) is selected, the program calculates the temperature drop as if it is a rectangular opening with open area equal to $Lhmax \cdot Lwmax$ (corresponding to opening factor 1.0).

In addition to the thermal properties, the zone volume must be input with the H/D/W option in `&-NET-ZONE`s and only one window to an external node may be defined.

As soon as the window to outside is opened, the actual zone temperature will be taken as the initial value for the thermal model. The zone temperature as defined in `&-NET-ZONE`s or in a room temperature schedule will be replaced with the time dependent zone temperature calculated with the thermal model.

Description:

Parameters	Description	Default value	Input format, range
Zone ID	starting with '*', followed by the identification of a defined zone	''	String
Conductivity	Mean value of thermal conductivities of all walls in the zone	0.0	Real
Density	Mean value of the densities of all walls in the zone	0.0	Real
Capacity	Mean value of the thermal capacities of all walls in the zone. Obs.: capacity has the unit [J/kg/K] !	0.0	Real
Wall Thickness	Mean value of the thickness of all walls in the zone	0.0	Real

&-NET-LINks (Net Links)

```
&-NET-LINks
```

Link ID (-)	Type Name (-)	Zone ID From (-)	Zone ID To (-)	Height From [m]	Height To [m]	Own Height Factor [-]	Act. Val. [-]	3Dflow or Press. [pres]	Schedule Name (10char) Ref Link Name [-]

Example

```
  5    CRxx   z1  z5    0.   0.     1.      1.
# Links L1, L2 and L3 form a T-junction
  L1   DS-1   -1   2    0    0      d       1        0        L2
  L2   DS-1   2   z3    0    0
# Link from zone 2 to a zone with 4 Pa
  L3   DS-1   2    4Pa  0    0
# Window with window schedule, fan with fan schedule
  L4   WI-1   4    5    0    0      1       1        0        W1
  L5   FA-1   5    6    0    0      1       1        0        F1
```

The zone identifier denotes

- ◇ internal zones,
- ◇ external nodes (characterized by a preceding minus sign), or
- ◇ nodes with constant pressure (a value immediately followed by the key characters 'Pa' denote the pressure level in Pascal).

A node with a constant pressure has a fixed wind pressure equal to the value given under `Zone ID`. For the total pressure, a stack pressure according to link height and ambient temperature is added.

Height relations are shown in figure 4.15. The absolute heights are the sum of the height of the zone reference plane plus link height in the respective zone. The absolute `From` and `To` height values are equal for horizontal links. If they are different, the link is not horizontal and the flow path is calculated as having a vertical component (e.g., a ventilation shaft).

NOTE Internal (i.e. zone) nodes are located in the zone reference plane. External nodes are located in the ground plane.

NOTE Window components may not have a vertical component. The program's internal tolerance for warning about differences in the absolute `From` and `To` height of the *window component* is set to 0.02 m.

Three links consisting of duct components (`&-DS`) which connect one node with three other nodes represent a T-junction (see figure 4.14). If the dynamic pressure loss due to converging or diverging flow in the T-junction has to be taken into account automatically, the three links have to be input according to the following conventions:

One part of the main (straight) duct must have the junction-node on its `To` side of the link. In the input line of this link, the `ID` of the link with the branch duct has to be given as reference link. (column `Ref Link Name`). The third link connecting the junction node is then interpreted as second main duct part. In the example, links L1 and L3 are the two main duct parts and link L2 is the related branch. Zone 'a2' is an auxiliary zone. In the current version of T-junction model, equal diameters for all of the three ducts and an angle of 90^o for the branch duct are assumed.

It is not allowed to have more than these three links to the central node (node 'a2' in the example).

NOTE T-junctions often cause convergence problems. If this happens, even is solver type 6 (see page 58) is chosen, it is recommended not to use it, but to input the dynamic pressure loss in the junction as

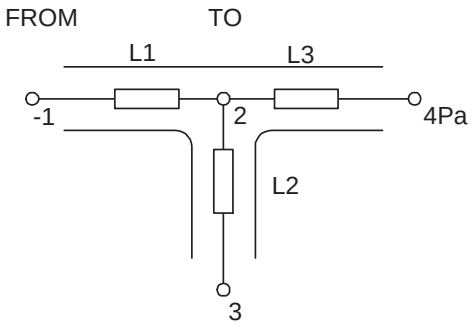


Figure 4.14: Definition of T-Junction nodes:
At the link which constitutes the main (straight) part of a T-junction node, the TO node represents the junction node. In the input line of this link, the branch link has to be referenced in column `Ref Link Name`. The figure corresponds to the example input.

Description:			
Parameters	Description	Default value	Input format, range
Link ID	Name of link	‘ ’	String 20 char
Type Name	Prefix and name of defined air flow component (without preceding ‘*’)	-	String
Zone ID From	`From` zone identifier (zone must be defined!)	‘ ’	String
Zone ID To	`To` zone identifier (zone must be defined!)	‘ ’	String
Height From	Height of link relative to `From` zone reference plane (see figure 4.15)	0.0	Real
Height To	Height of link in relative to `To` zone reference plane (see figure 4.15)	0.0	Real
Own Height Factor	Cosine of the angle between opening plane and vertical plane for slanted windows (see section `&-WI`, page 80 and figure 4.8).	1.0	Real 0.0 . . . 1.0
Act. Val.	Actual Value for Crack (s. page 62) Actual RPM or Factor for Fan (s. page 65) Opening Factor for Window or Door (s. page 80) Actual Scale Factor for Test Data (s. page 87) Factor for Zeta value for Duct (s.page 68)	1.0	Real
3Dflow or Press.	(not implemented)	0.0 Pa	Real
alternatively, if applicable:			
Schedule Name	Name of schedule applicable to the air flow component. The first character of the name indicates the schedule type: W : Window schedule F : Fan schedule	‘ ’	String 10 char
Ref Link Name	Reference link for T-junction or for Related Flow component (link must be defined!)	‘ ’	String

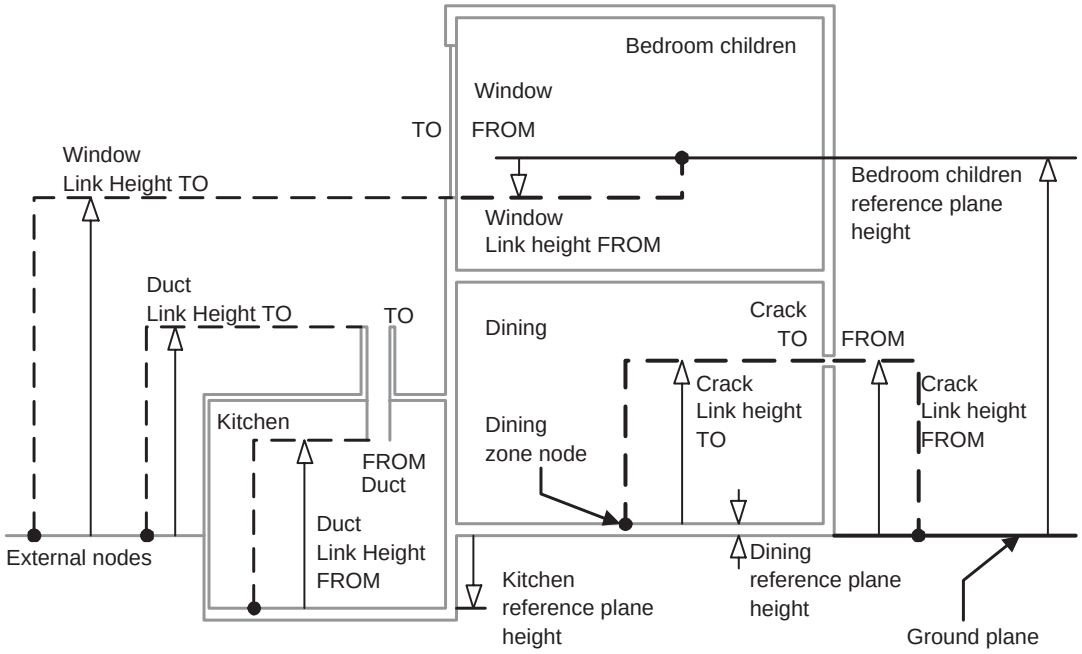


Figure 4.15: Definition of link height. Example zones from `&-NET-Zon`, see page 92 (Links not reproduced in example).

fixed coefficients (zeta values) with the corresponding 'DS' components, according to an estimated relationship between the flows in the junction.

NOTE A zone cannot be linked only with 'RF' components, there must also exist some normal links, like a crack or a fan.

Kitchen Hood Description under &-NET-LINks

The following input under `&- NET-LINks` is required for the kitchen hood modelled with an unbalanced related flow component according to figure 4.12 on page 91

Example `&-NET-LINks`

Link	Type	Zone ID		Height		Height Factor	Act. Val.	3Dflow or Press.	Sched Name (10char)
ID	Name	From	To	From	To				Ref Link Name
(-)	(-)	(-)	(-)	[m]	[m]	[-]	[-]	[pre]	[-]
L1	CR1	-1	kit	1.1	1.1				
L2	CR2	kit	hoo	1.5	1.5				
L3	FA1	hoo	-2	3	3	d	800	d	F1
L4	RF1	hoo	kit	2.5	2.5	d	d	d	L3

The fan has a normal speed of 800 rpm and a schedule F1, which switches the fan in different positions. To be able to use link L3 as reference, it must appear before the RF that is using it, or else the reference flow rate would not be known or contain an old value at the time the related flowrate is calculated.

It is recommended to use Related Flow types always in pairs to have balanced flow, like in the next example.

Example `&-NET-LINks`

Link	Type	Zone ID		Height		Height Factor	Act. Val.	3Dflow or Press.	Sched Name (10char)
ID	Name	From	To	From	To				Ref Link Name
(-)	(-)	(-)	(-)	[m]	[m]	[-]	[-]	[pre]	[-]
L1	CR1	-1	kit	1.1	1.1				
L2	CR2	kit	hoo	1.5	1.5				
L3	FA1	hoo	-2	3	3	d	800	d	F1
L4	RF1	hoo	kit	2.5	2.5	d	d	d	L3
L5	RF1	kit	hoo	2.5	2.5	d	d	d	L3

This results in flows in both directions and will have no effect on the flow balance during the iteration process. It will result in the desired flow exchange that can spread pollutants (see page 89).

4.3 Schedules

&-SCH-WINdow (Window Schedules)

```
&-SCH-WINdow schedules
 ------------------------------------------
| Schedule  |    Time             | Opening  |
| *WName    |                     | Factor   |
|  (-)      |    (-)              |   (-)    |
|___________|_____________________|__________|
```

Example

```
    *W1            06:00            .50
                   06:10           1.000
                   06:15            .0
                   08:00            .10
                   12:00            .30
                   18:40            .0
    *W2        FRI_06:00           1.00
               FRI_07:00            .0
                   07:00            .40
                   08:00            .0
```

or if the data are defined in a separate file:

```
F: W3  window1.dat
```

With the window schedule the opening factor of a window can be changed. The schedule name must be specified in the last column of the section `&-NET-LIN`ks in the lines with the corresponding windows. The value in column "Actual Value" of this line will then be replaced by the value given in the schedule.

Description:

Parameters	Description	Default value	Input format, range
Schedule *WName	Name of schedule or file identifier (incl. path). Name must be preceded by '*W'. Data file assignment format: F: W<Name> <file_name.ext>	-	Name: String (2+) 8 char File: String 160 char
Time	Date and / or time	-	Date_time string
Opening Factor	Opening factor of window (see window definition page 80)	-	Real 0.0 . . . 1.0

&-SCH-FANs (Fan Schedules)

```
&-SCH-FAN schedules
```

Schedule *FName (-)	Time (-)	Fan Speed Factor (-)

Example

```
   *F1        FRI                 1.1
              SAT                 0.8
              SUN                 0.75
              MON                 1.4
   *F2           06:00            1.0
                 07:00            0.7
                 18:00            1.18
   *F3        WND                 1.0
              WDY                 0.5
```

or if the data are defined in a separate file:

```
   F: F3     Fan3.Dat
```

Similar to the window schedule, the fan schedule is connected to the air flow component by specifying the schedule name in the corresponding line under `&-NET-LIN`ks. The input in column "Actual Value" under `&-NET-LIN`ks will be multiplied with fan speed factor.

Description:

Parameters	Description	Default value	Input format, range
Schedule *FName	Name of schedule or file identifier (incl. path). Name must be preceded by '*F'. Data file assignment format: F: F<Name> <file_name.ext>	-	Name: String (2+) 8 char File: String 160 char
Time	Date and / or time	-	Date_time string
Fan Speed Factor	Multiplication factor for actual fan speed specified in "Actual Value" in `&-NET-LIN`ks section	-	Real ≥ 0.0

&-SCH-TEMperature (Temperature Schedules)

```
&-SCH-TEMperature schedules
 ------------------------------------------
| Schedule  |    Time            |Temperature|
| *TName    |                    |           |
|  (-)      |    (-)             |  (temp)   |
|___________|____________________|___________|
```

Example

```
*T1        SAT                 17
           MON                 20
```

or if the data are defined in a separate file:

```
F: T2     Temp.Dat
```

Temperature values from a schedule replace the present temperature values for all zones with the respective schedule name in the last column of the &-NET-ZONes description

Description:			
Parameters	Description	Default value	Input format, range
Schedule *TName	Name of schedule or file identifier (incl. path). Name must be preceded by '*T'. Data file assignment format: F: T<Name> <file_name.ext>	-	Name: String (2+) 8 char File: String 160 char
Time	Date and / or time	-	Date_time string
Temperature	Room temperature value	- (^{o}C)	Real

&-SCH-HUMidity (Humidity Schedules)

```
&-SCH-HUMidity schedules
```

Schedule *HName (-)	Time (-)	Humidity (humi)

Example

```
*H1          07:00          12
             07:40          10
             08:00          11
```

or if the data are defined in a separate file:

```
F: H2     Hum2.Dat
```

Humidity values from a schedule replace the present humidity values for all zones with the respective schedule name in the last column of the &-NET-ZONes description

NOTE The humidity must be set to zero in all zones if it is defined as pollutant under &-POL-DES and under &-NET-ZP, and transport mechanism shall be studied. For details, see page 36.

Description:

Parameters	Description	Default value	Input format, range
Schedule *HName	Name of schedule or file identifier (incl. path). Name must be preceded by '*H'. Data file assignment format: F: H<Name> <file_name.ext>	-	Name: String (2+) 8 char File: String 160 char
Time	Date and / or time	-	Date_time string
Humidity	Humidity in a zone	- (g/kg)	Real >=0.0

&-SCH-SINk (Sink Schedules)

```
&-SCH-SINk schedules
```

Schedule *SName (-)	Time (-)	Sink Factor (-)

Example

```
*S-H2O        12:00          0.5
              18:00          0.0
*S-20-H2O     20:00          1.2
              00:30          0.3
```

or if the data are defined in a separate file:

```
F: S-CO2  Sink.Dat
```

The sink schedule name has to start with '*S', followed by an optional combination of digits and separators, followed by the first unambiguous part of the name of the pollutant. The optional part may be used to make more schedules for one and the same pollutant.

The format for the sink schedule name is *S[number]polname

*S	mandatory characters
[number]	optional sequence of digits and separators ('-', '_', '.', ':', '[', ']')
polname	first unambiguous part of the pollutant name

A few examples are: `Spol1`, or `SCO`, or `S23pol1`.

For each zone for which the schedule name is found in the last column of the keyword `&-NET-ZONes`, the value of the corresponding sink strength in the zone pollutant description, `&-NET-ZP`, is changed. The sink strength is multiplied with the sink factor given in the sink schedule.

Description:

Parameters	Description	Default value	Input format, range
Schedule *SName	Name of schedule or file identifier (incl. path). Name must be preceded by '*S'; it must be followed by either a combination of numbers and separators like '-' or '_', followed by the first unambiguous part of the name of the desired pollutent, or by the first unambiguous part of the name of this pollutant directly. Data file assignment format: F: S<Name> <file_name.ext>	-	Name: String (2+) 8 char File: String 160 char
Time	Date and / or time	-	Date_time string
Sink Factor	Factor of actual sink strength to value given in zone pollutants data block (`&-NET-ZP` on page 96)	-	Real ≥ 0.0

&-SCH-SOU rce (Source Schedules)

```
&-SCH-SOUrce schedules
```

Schedule *QName (-)	Time (-)	Source Factor (-)

Example

```
   *Qh2o          11:40            1.0
                  12:40            0.1
   *Q-CO2         00:00            2
                  09:00           40
                  12:00            2
```

or if the data are defined in a separate file:

```
   F: QCO2   OccSou.Dat
```

The source schedule name has to start with '*Q', followed by an optional combination of digits and separators, followed by the first unambiguous part of the name of the pollutant. The optional part may be used to make more schedules for one and the same pollutant.

The format for the source schedule name is *Q[number]polname

*Q	mandatory characters
[number]	optional sequence of digits and separators ('-', '_', '.', ':', '[', ']')
polname	first unambiguous part of the pollutant name

A few examples are: `Qpol1`, or `QCO`, or `Q23pol1`.

For each zone for which the schedule name is found in the last column of the keyword `&-NET-ZONes`, the value of the corresponding source strength in the zone pollutant description, `&-NET-ZP`, is changed. The source strength is multiplied with the source factor given in the source schedule.

Description:

Parameters	Description	Default value	Input format, range
Schedule *QName	Name of schedule or file identifier (incl. path). Name must be preceded by '*Q'; it must be followed by either a combination of numbers and separators like '-' or '_', followed by the first unambiguous part of the name of the desired pollutant, or by the first unambiguous part of the name of this pollutant directly. Data file assignment format: F: Q<Name> <file_name.ext>	-	Name: String (2+) 8 char File: String 160 char
Time	Date and / or time	-	Date_time string
Source Factor	Factor of actual source strength to value defined in zone pollutant data block (`&-NET-ZP` on page 96)	-	Real ≥ 0.0

&-SCH-OCC upant (Occupant Schedules)

```
&-SCH-OCCupant schedules
 ------------------------------------------------------------------------
|Schedule   |    Time            |   Zone  |Activity Level | Number of |
|*OCCn      |                    |   ID    |Factor         | Occupants |
| (-)       |    (-)             |   (-)   |   [-]         |    [-]    |
|___________|____________________|_________|_______________|___________|
```

Example

```
   *OCC1          07:00            z1              0.75           1
                  07:00            z6/z7           1.00           0
                  07:30            z2/z3/z5        1.00           5
                  07:50            z9/z4/z8        1.00           2
                  08:00            z1-z9           0.75           4
                  12:00            z1-z9           1.00           0
                  13:00            z6/z7           0.75          28
```

or if the data are defined in a separate file:

```
F: OCC2   Occup.dat
```

The name of the schedule determines which occupant type the schedule entry refers to. The schedule name must be 'OCC1' to 'OCC5' (preceded by '*') which means that the schedule data belongs to occupant type identified by number 1 to 5.

Without occupant schedule, an occupant type defined in data section `&-OCCUPANt` is not attached to any zone. An occupant type can be absent or be present in one or more zones during a timestep. More precisely, the number and the activity level of an occupant type in the zones may vary. If an occupant shall be absent in a certain zone after that occupant had been present, the number of occupants of that type must be explicitly set to zero. In the example, occupants of type one are 'removed' from zones z6 and z7 at 7:00, and from zones z1 – z9 at 12:00. Moving two occupants to zones z9, z4 and z8, respectively at 7:50 does not change the occupant number of any other zone. Never setting an occupant number for a zone is equivalent to setting the number of occupants of that type to zero. I.e., if a zone z10 would exist in the example, it never would be occupied by any occupant.

Because more than one zone at a time can be affected, lists of zone identifiers can be specified.

Description:

Parameters	Description	Default value	Input format, range
Sched. *OCCn	Name of schedule or file identifier (incl. path). Name must be preceded by '*OCC'. Allowed names are 'n', where $1 \leq n \leq 5$ is the occupant type number. Data file assignment format: F: OCC<n> <file_name.ext>	-	Name: String (2+) 8 char File: String 160 char
Time	Date and / or time	-	Date_time string
Zone ID	(List of) Zone Identifier(s) of zone(s) in which the occupant is located. Zone must be defined (ranges are allowed).	-	
Activity Level Factor	Factor of actual activity to the one specified in the occupant description data section.	1.0	Real ≥ 0.0
Number of Occupants	Number of occupants	1	Integer ≥ 0

This allows to change the presence of an occupant type simultaneously for more than one zone. Conversely, occupants may change presence and activity differently in different zones at the same time. Therefore, more than one entry with the same date and time may be given in the occupant schedule. An example of occupancy and occupant schedule can be found under the factory example on pages 133ff.

The activity level of an occupant type (defined under `OCCUPANt`) is determined by the activity level factor given in the respective schedule line, but only for the zones specified in the schedule. The number of occupants of an occupant type is set to the number of occupants given in the respective schedule line, but only for the zones specified in the schedule. The (additional) source strength of pollutants due to the presence of occupants is the product of number of occupants, activity level factor and the nominal source strength defined under `&-OCCUPANt`.

The activity level of an occupant type may be set to zero. If the number of occupants of that type in the respective zone is greater than zero at the same time, the occupant type acts like a sensor: it does not contribute to the source strength in the zone. However, this type is exposed to the possible pollution due to other pollutant sources or even other quantities like air flow in the zone. This exposure then may be evaluated with an appropriate histogram output. For details, see general remarks on pollutants on pages 36ff and definition of histograms, especially for occupants, on pages 40ff.

Values for number of occupants or both number of ocupants and activity level factor or may be left blank in the schedule. Then their default values are assumed during the next time step. The default values are 1.0 for the activity level factor and 1 for the number of occupants.

&-SCH-METeo Data (Meteo Data Schedule)

```
&-SCH-METeo data
  _________________
 |1.| Dataset Name |
 |__|______________|
  ____________________________________________________________________________
 |2.|   Time       |       Wind       |Temperature |Humidity |Barometric |
 |__|              |__________________|            |         |Pressure   |
 |                 | Speed  |Direction|            |         |Absolute   |
 |(-)              |(velo)  |  (deg)  |  (temp)    |[humi]   |  [kPa]    |
 |_________________|________|_________|____________|_________|___________|
```

Example

```
Meteoname
1968may23_08:30    2.0        10        15.1         8.4      98.80
1968may23_12:45    0.          0.       20.         10.      101.325
```

or, if file assigned:

```
F: berk87.met COMIS
```

The meteo data can be stored under the keyword `&-SCH-MET` in the CIF-file or in a separate weather file. When using a weather file, the data under the first header of the meteo schedule must contain a data file assignment, marked by an 'F:' followed by the respective filename and a keyword for the meteo data file format. When both, separate weather file and meteo description in the CIF-file, are used and there are entries for the same time in both files, the CIF meteo description 'overwrites' the weather file data.

NOTE If schedule data are entered in the CIF-file, data set name is required.

There is a conversion utility COMISweather available which can be downloaded from the COMIS eGroups homepage,

`http://www.egroups.com/files/comis/`

Weather files of different formats can be converted to COMIS format with this tool.

Description:

Parameters	Description	Default value	Input format, range
Dataset name	Any name		String 80 char
	For data file assignment: F:<filename> <format>		String, <format> = COMIS or DOE2
Time	Date and time	jan01_	Date_time string
Wind Speed	Wind speed at meteo station reference height	0.0 m/s	Real > 0.0
Wind Direction	Wind direction at meteo station reference height	0.0	Real 0.0 . . . 360
Temperature	Dry bulb temperature at meteo station	20.0 ^{o}C	Real
Humidity	Absolute humidity at meteo station	0.0 g/kg	Real
Pressure	Absolute barometric pressure at meteo station[a]	101.325	Real > 0.0

[a] The barometric pressure must be in kPa.

Two formats are allowed for the weather file.

1. COMIS format (default)

The data format for CIF schedule input can be used in the weather file, too. That is, every data line starts with a date_time string followed by the information about wind speed, wind direction, etc. If there are data missing, the program assumes default values.

Example
```
1991may23_08:30:10  2.0 10  15.1    8.4 98.80
1991may23_09:35:30  1.8 11  15.1    8.4 97.00
```

If many sets of weather data in equal time steps are used, a more convenient way of inputting the data is to specify the start time and the time interval between the data lines in the first line of the meteo data file. The first character of this line must be an asterisk ('*') to mark that this kind of format is used. It is not necessary to specify both parameters, default values are: start time = Jan 01 and hourly data changes.

Examples
```
*1991may23_10:00    3600
2.0 10  15.1    8.4 98.80
2.3 11  16.0    8.4 97.00
```
or
```
*1991jun01_
2.0 10  15.1    8.4 98.80
2.3 11  16.0    8.4 97.00
```
or
```
*
2.0 10  15.1    8.4 98.80
2.3 111 6.0 8.4 97.00
```
The weather data file may contain blank lines, which are ignored.

Since the weather file is sequentially read by the program, the time statement must at least consist of month and day specification. If the year is omitted, the year of the start of the simulation interval is assumed. A time statement only containing a time and no date cannot be processed in that way that the program repeatedly reads the data of this line for every day which shall be simulated.

2. DOE2 format

It is also possible to use weather files in the (packed) DOE-2 format. This has the advantage that not two different weather files need to be stored if the user runs both, DOE-2 and COMIS simulations (DOE-2 is a thermal simulation program).

The program expects the weather data to be in the DOE-2 format when the keyword "DOE2" is specified after the filename in data block 1.

Example
```
F: zagreb.bin DOE2
```

&-SCH-POLlutant (Outdoor Pollutant Concentration Schedule)

```
&-SCH-POL outdoor concentration data

 _________________
|1.|DATASET NAME  |
|__|______________|

 ____________________________________________________________________
|2.| Time            |     Pollutant  Concentration                    |
|__|                 |_________________________________________________|
|                    |  No1    |  No2    |  No3    |  No4    |  No5    |
|    (-)             | (conc?) | (conc?) | (conc?) | (conc?) | (conc?) |
|____________________|_________|_________|_________|_________|_________|
```

Example

```
Typical_Pollutant_1
  0101_00:00          0.  0.  0.  0.  0.
```

Or in case of a pollutant file assignment:

```
F:  examp88.poll
```

The concentrations given here are global outdoor values for the pollutants no 1 to 5, if defined under `&-POL-DES`. For every external node, individual factors can be defined under `&-NET-EXT`ernal node. The schedule concentration of pollutant no. 1 is multiplied with the outdoor concentration factor (third column of `&-NET-EXT`ernal) of the nodes. The same is done with the pollutants no. 2 to 5.

If no concentration is found for a pollutant in the pollutant schedule, 0.0 is taken as default.

The keyword `meteo` in the column of the pollutant 'H2O' may be used to indicate that the water vapour concentration shall be taken from the meteo schedule.

NOTE If the keyword `SCH-POL` is missing or has no data, and if pollutant 'H2O' is defined, the concentration values for 'H2O' are taken automatically from the meteo schedule.

This means that it is necessary to set the outdoor concentration of 'H2O' explicitly to zero with this schedule if humidity input under `&-NET-ZON` (and `&-NET-ZL`, `&-SCH-HUM`) is used.

How pollutant calculation routines are activated is described under data section `&-NET-ZP` on page 96.

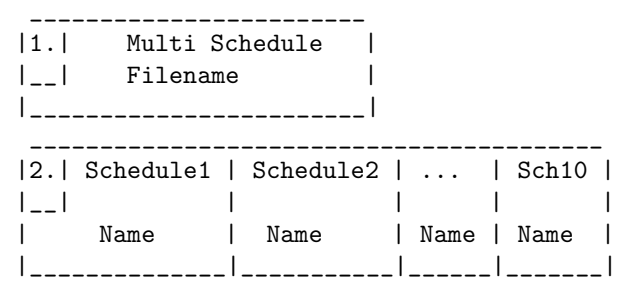
NOTE If schedule data are entered in the CIF-file, data set name is required.

Description:

Parameters	Description	Default value	Input format, range
Dataset name	Any name. Default: 'OutsidePollutant' For data file assignment: F:<filename>	(see description)	String 80 char
Time	Date and time	jan01_	Date_time string
Pollutant Concentration	Global outdoor pollutant concentration for pollutants no 1 dots 5; the number of concentration values per line must be equal to or or larger as the number of pollutants defined	0.0	Real ≥ 0.0

&-SCH-MULti schedule data (Multi schedule data)

```
&-SCH-MULti schedule data
```



Example
```
F: multi.dat
      t.house     INVALID     t.attic
```

The multi schedule is used to avoid having a separate file for each schedule. Entries in the multi schedule have to be sorted in ascending order of time. The values of the last data line in a multi schedule are valid for the rest of the simulation, irrespective of the interval length until the stop time of the simulation. No cyclic 'reuse' of schedule data is possible for multi schedule data.

No sorting needs to be done internally by the Time Management System during the program run if no other schedule is used.

Schedules supported by multi schedule are:

Window (W), Fan (F), Temperature (T), Humidity (H), Sink (S), Source (Q), Meteo (MET) and Pollutant (POL).

The schedule names have to correspond to the conventions defined in the preceding chapters. The predefined characters of the schedule names are given in brackets. However, the meteo schedule spawns over several columns in the multi schedule file:

Wind speed Wind direction Temperature Humidity Pressure

The Pollutant Schedule uses the names 'pol1' to 'pol5' for the outdoor pollutants no 1 to 5.

If the name is 'INVALID' the column is skipped.

The multi schedule file has a form similar to the COMIS weather file (see section SCH-METeo data on page 112). In the first line is an asterisk, optionally followed by the start time and date and the interval length in seconds. It is not necessary to specify both parameters when one wants to accept the defaults: start time = Jan 01 and hourly data changes.

The next line gives the names of the schedules. These have to match the names in the CIF-file ('INVALID' in the CIF-file matches any name, however). After this header the schedule data follow.

Description:

Parameters	Description	Default value	Input format, range
Filename	Name (incl. path) of the Multi Schedule File F: <file name.ext>		String 160 char
Name	Name of the schedule in the appropriate column. The type of the schedule is determined by the name. If the name is 'INVALID' the column is skipped. Names are case sensitive.		Name: String 10 char. Line (names incl. spaces): 200 char Max. 10 columns

Example

```
*   1992jan01_0:00    3600
t.house pol1    t.attic
20.1     .1E-4     18.3
23.4     .3E-4     19.1
```

NOTE Only one multi schedule file is allowed per input file.

4.4 Building Description

&-CP-BUILding (Building Reference Height)

```
&-CP-BUILding reference height for Cp data
  ----------
 | Height   |
 | (m)      |
 |----------|
```

Example

```
10.0
```

C_p values for a specific building relate the wind pressure at the facade to wind speed and direction at a certain height above ground. This building reference height, usually the eaves height of the building (see figure 4.17 on page 122), is input here.

In COMIS, the facade is divided into facade elements. C_p values for facade elements are input under `&-CP-VALUe`s. With that, the wind pressure at the external nodes, defined under `&-NET-EXT`ernal nodes, is also known. Wind data, given in `&-SCH-METe`o data, from the meteorology station, described under `&-ENV-WIN`d, is converted to wind data at building reference height.

In case of low buildings in rough terrain and large surrounding obstacles, Cp data from wind tunnel or on site measurements should be used.

Description:			
Parameters	Description	Default value	Input format, range
Height	Reference height for wind data to which building cp-values are related	10.0	Real > 0.0

&-CP-VALUes (Cp Values)

```
&-CP-VALUes

  -----------------
 |1.|Dataset Name  |
 |__|______________|

 ------------------------------------------------------------------------------
 |2.| *      | Wind Direction with respect to Building Axis   (first line ) |
 |__| Facade | Cp Values                          (second and following lines) |
 | Elem.Name |      |      |      |      |      |      |      |      |      |
 |           |(deg) |[deg] |[deg] |[deg] |[deg] |[deg] |[deg] |[deg] |[deg] |
 |   (-)     | (-)  | [-]  | [-]  | [-]  | [-]  | [-]  | [-]  | [-]  | [-]  |
 |___________|______|______|______|______|______|______|______|______|______|
```

Example

```
Example_House
    *         0      90     180    270
    1       -.5      .8     -.5    -.8
```

or

```
My_House
    *       0        90     180    270
    4       1.0      -.49   -.45   -.49
    2       -.49     1.0    -.49   -.45
    3       -.45     -.49   1.0    -.49
    1       -.49     -.45   -.49   1.0
    *       351     352     353
    4       -.49     -.45   -.49
    2       1.0      -.49   -.45
    3       -.49     1.0    -.49
    1       -.45     -.49   1.0
```

or if data file assignment:

```
F: leso.cp1
```

Wind pressure coefficients (Cp values) belonging to a certain facade element can be given for several wind directions. The angle of the wind direction is measured clockwise relative to the building axis, see figure 4.16. The building axis can be defined as fits best to the project.

More than one data set may be given, thus allowing for more wind direction columns. The facade element names must be the same in each data set. Each data set can have a maximum of eight

Description:

Parameters	Description	Default value	Input format, range
Dataset name	Dataset name For data file assignment: F:<filename>	'cpname'	String 80 char
Facade Elem.name	First line: '*', next lines: Name or number of facade element to which Cp values in data line belong	-	String 20 char
Wind Direction with respect to Building Axis	Wind direction is given relative to the building axis (measured clockwise). The first data line contains the wind direction values. 0^o : against direction of building axis	0.0	Real 0.0 . . . 360.0
Cp Values	Wind pressure coeffficients for wind directions given in the first data line	0.0	Real

wind direction columns.

NOTE If schedule data are entered in the CIF-file, data set name is required.

C_p values can be obtained from different sources. The most important sources are listed below.

1: Literature: "rough-and-ready" approximate values of C_p are conveniently tabulated in many publications – such as the highly recommented AIVC Technical Note 44 [1], for different shapes and sizes of buiding. Technical Note 27 [2] is more appropriate for C_p values of dwellings with a pitched roof. The AIVC publish other technical reports that may also be of interest (see http://www.aivc.org/publications.html). ASHRAE Fundamentals also has some data, but this is not as comprehensive.

2: Computer programs using data bases (Examples): When given the building geometry, these calculate C_p values according to a correlation from a database of measurements. However, we can not guarantee that such programs provide more accurate C_p values than conventional tabulated data. None of these programs are sold commercially yet, but you can use them now by contacting the developers.

- ◇ "CPCALC+" contact assoc. prof. Mario Grosso, School of Architecture, Polytechnic University of Turin, Viale Mattioli 39, 10125 Torino, Italy.
- ◇ "Cp-Generator" developed at TNO applied research organisation. See publication Airbase NO 9060 entitled: Pressure simulation program. by Knoll B, Phaff J C and de Gids W F. (Proceedings 16th AIVC conference). See also AIVC's Air Infiltration Review Vol. 17, No. 3, june 1996. Since these publications came out, major improvements have been made to Cp-generator. Also validations have been carried out and are still going on. It even has output formatted for COMIS.

3: Detailed modelling with specialized CFD software: computational fluid dynamics (CFD) modelling provides almost as good results as wind-tunnel measurements.

4: Wind-tunnel measurements: these are the most reliable data.

4.5 Environment Description

&-ENV-BUIlding (Building Related Parameters)

```
&-ENV-BUIlding related parameters
 ------------------------------------------------------------
|1.| Altitude |Angle Building |   Geographic Position      |
|__|          |_______________|____________________________|
|             |North to Axis  |Latitude +=N | Longitude +=E|
|      (m)    |  (deg)        | [deg]   -=S |  [deg]    -=W|
|_____________|_______________|_____________|______________|
```

Example

```
588.0            34.04          7.65          10.0
```

The building axis can be defined as fits best to the project. Its angle is measured clockwise from north, see figure 4.16.

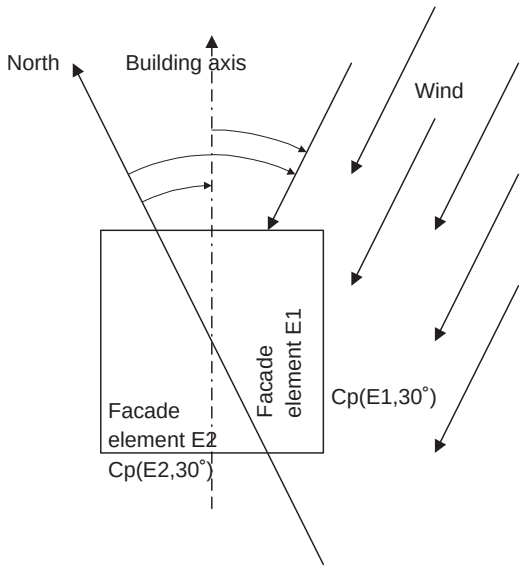


Figure 4.16: Building orientation: measured clockwise from north to Building axis.
Cp values for each facade element can be given for several wind directions.
Wind directions given under &-CP-VALUes are measured clockwise from the building axis.
Wind directions given under &-ENV-WINd are measured clockwise from north.

Description:			
Parameters	Description	Default value	Input format, range
Altitude	Altitude of ground plane above sea level	0.0	Real
Angle Building	Angle from north to the building axis, measured clockwise (see figure 4.16); 0^o : north	0.0	Real 0.0 . . . 360.0
Latitude	Latitude $> 0^o$: northern hemisphere Latitude $< 0^o$: southern hemisphere	43.0	Real -90.0 . . . 90.0
Longitude	Longitude $> 0^o$: east Longitude $< 0^o$: west	0.0	Real -180.0 . . . 180.0

&-ENV-WIN d (Wind and Meteo Related Parameters)

```
&-ENV-WINd and meteo related parameters

 ------------------------------------------------------
|1.|  Ref. Height  |  Altitude     | Wind Velocity    |
|__| for Wind Speed| Meteo Station | Profile Exponent |
|                  |               | Meteo Station    |
|       (m)        |    (m)        |   (prof)         |
|__________________|_______________|__________________|

 ---------------------------------------
|2.| Wind Direction | Wind Velocity     |
|__|    Angle       | Profile Exponent  |
|                   | Building Location |
|       (deg)       |     (prof)        |
|___________________|___________________|
```

Example

```
10.0              456              0.14
 0              .22
 45             .16
 90             .33
 175            .28
```

COMIS supports the two main approaches for wind velocity profiles, the power law profile and the logarithmic profile. At the keyword `&-PR-UNITS` (see page 49) the profile to be used can be chosen. Default is power law profile.

No matter which profile is taken, the validity for urban areas is usually small, unless the building is taller than its surrounding. In case of low buildings in rough terrain and large surrounding obstacles, direct C_p data from wind tunnels or on site measurements should be used.

From this it is clear that the wind velocity at building reference height (see `&-CP-BUIL`) is not a direct input.

The program uses the given wind speed at the meteo site in reference height to calculate the speed above the boundary layer using the wind velocity profile of the meteo station. The boundary layer is 60 m high for smooth terrain. In rough terrain, the height of the boundary layer is increased. For the power law profile, the condition for rough terrain is $\alpha > 0.34$, and for the logarithmic profile, $z0 > 1.5$.

Description:			
Parameters	Description	Default value	Input format, range
Ref. Height for Wind Speed	Height above ground for wind data at the meteo station	10.0	Real $1.0 < h_m < 400.0$
Altitude Meteo Station	Altitude of meteo station above sea level	0.0	Real
Wind Velocity Profile Exponent Meteo Station	Parameter of the wind mean velocity profile at meteo station. Exponent α in power law function or z0 for logarithmic profile.	0.14	Real $0.085 < \alpha \leq 0.87$, $0.0002 < z0 \leq 40.0$
Wind Direction Angle	Angle of wind direction measured clockwise from north, see figure 4.16	0.0	Real 0. . . . 360.
Wind Velocity Profile Exponent Building Location	Parameter of the wind mean velocity profile at building location. Exponent α in power law function or z0 for logarithmic profile.	0.18	Real $0.085 < \alpha \leq 0.87$, $0.0002 < z0 \leq 40.0$

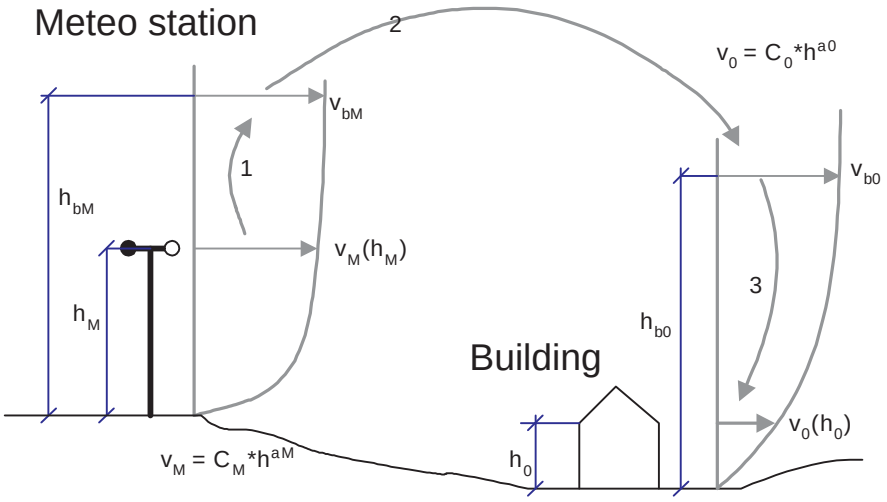


Figure 4.17: Wind Velocity Profile: relations at meteo and at building site, example for power law profile.
Indices: M Meteo station, 0 building site, b boundary. h_0 Building reference height defined under `&-CP-BUIL`, h_M Reference height at meteo station

The wind speed above the boundary layer at the meteo station is assumed to be equal to the wind speed at the same height above ground at the building location. With the wind profile near the building the velocity at the building reference height defined in `&-CP-BUIL`ding is calculated.

The following table [27] gives comparable values of $z0$ and α and the respective values for the height above ground of the boundary layer (*zbound*).

$z0$ (m)	α (-)	*zbound* (m)	terrain	(according to meteorological definitions)
0.0002	0.085	60	sea	
0.005	0.118	60	flat	no obstacles, beach, ice plain, snow
0.01	0.128	60		
0.03	0.149	60	open	low grass, field without crop (fallow land)
0.07	0.171	60		
0.10	0.182	60	roughly open	low crops, low hedges, few trees, very few h
0.25	0.218	60	rough	high and low crops, large obstacles at distances
				15*H, dense rows of trees, low orchards
0.5	0.257	60	very rough	obstacles at distances of 10*H, spread wood
				farm buildings, vineyards
1.0	0.313	60	closed	obstacles at distances of less than a few times H,
				large woods, villages small cities
2.0	0.377	80	city centre	alternated low and highrise, old woods with
5.0	0.456	200		trees alternated with frequent open areas
10.0	0.542	400		
20.0	0.668	800		
40.0	0.869	1600		

Reference values for α:

0.10 : level surface with very small obstructions, grassland;

0.22 : rolling or level surface broken by numerous obstructions, such as trees or small houses;

0.32 : heterogeneous surface with structures larger than one story

4.6 Pollutant Data

&-POL-DES cription (Pollutant Description)

```
&-POL-DEScription
  ------------------------------------------------
 | No  | Name              | Molar Mass | Decay   |
 | (-) | (-)               |  (g/mol)   |  [1/s]  |
 |_____|___________________|____________|_________|
```

Example

```
 1     pollutant_x          28.6
 2     CO2                  44
 3     FORMALDEHYD          30
```

Pollutant names may be longer than the fraction used for referencing in source and sink schedule names (see &-SCH-SINk on page 108, &-SCH-SOUrce on page 109) and at the occupant definition (see &-OCCUPANt on page 110).

How pollutant calculation routines are activated is described under data section &-NET-ZP on page 96.

NOTE Pollutants have to be numbered continuously starting with 1. Defining three pollutants with number 1, 3, 5 will lead to errors!

Molar Mass data is only used when needed according to how the source/sink strengths and concentration units are defined. Molar Mass has no effect for the pollutant 'ODOUR' that only may be used with units 'olf' and 'decipol'.

The decay constant r_k is defined as follows:

$$\frac{dm}{dt} = -r_k \cdot m$$

Description:

Parameters	Description	Default value	Input format, range
No	Number of pollutant; continuous numbering, beginning with 1.	1	Integer 1 . . . 5
Name	Name of pollutant; a pollutant name must start with a character A..Z,a..z	'pollutantx'	String 30 char
Molar Mass	Molar mass of respective pollutant	28.6 g/mol	Real > 0.0
Decay	Pollutant specific decay constant due to e.g. radioactivity	0.0 1/s	Real ≥ 0.0

&-POL-FICtive Sources (Pollutant Fictive Sources)

```
&-POL-FICtive source
 ------------------------------------------
|       | Pollutant |  Fictive   | Dependency |
| No    | name      |  Source    | indicator  |
| (-)   |   (-)     | [sour?/dep]|   [-]      |
|_______|___________|____________|____________|
```

Example

```
    1          CO2     1.44E-7      2
#          defined in                0  no dep
#          &-POL-DES                1-3  Room dep
#                                   5-6  Occ dep
```

Fictive sources are used for the determination of the effective flow in a zone. For the definition and use of fictive sources, please refer to section on Effective Flow rates on page 38.

The following table shows the meaning of the possible values for the dependency indicator:

Dependency Indicator	Description	Dependency unit [dep]
0	no dependency	1
1	zone floor area dependency	[m^2]
2	zone wall area dependency	[m^2]
3	zone volume dependency	[m^3]
5	occupant number dependency	[person]
6	average occupant activity dependency	[W/m^2]

The wall area includes the area of floor, ceiling and walls.

NOTE The zone volume must be input in the H/D/W format at `&-NET-ZON` because these values are used to calculate wall and floor area.

The average occupant activity is the mean activity of all occupant types in the zone under consideration:

$$< activity >= \frac{\sum_{types} occ_num \cdot occ_act}{\sum_{types} occ_num}$$

Description:

Parameters	Description	Default value	Input format, range
No	Number of fictive source	-	Integer
Pollutant Name	Name of pollutant defined in `&-POL-DES`cription	'pollutantx'	String 30 char
Fictive Source	Source strength of fictive source	0.0 kg/s/[dep]	Real
Dependency indicator	Indicator for dependency of source strength on room or occupant parameters	0	Integer 0 ... 3, 5,6

4.7 Occupant Data

&-OCCUPANt (Occupant Description)

```
&-OCCUPANt description

 ------------------------------------------------------------
|*No | Sex  | Age | Height| Mass| Metabolic| Cigarets| Name |
|    |      |     |       |     |  Rate    |         |      |
| (-)| (-)  | (a) |  [m]  | [kg]|  [W]     |  [1/h]  | (-)  |
|____|______|_____|_______|_____|_________ |_________|______|

 ------------------------------------------------------------------------------
| Pol Name| Source strength| Pol Name| Source strength| Pol Name| Source strength|
|   [-]   |      [sour?]   |   [-]   |      [sour?]   |   [-]   |      [sour?]   |
|_________|________________|_________|________________|_________|________________|
```

Example

```
*1    MIX    20.   1.7     70.   58.      0.0    adult
 CO2            d             H2O         5.5E-6
*2    MALE    5    1.03    15    60       0    child
```

Occupants may contribute to the source strength of certain pollutants in zones. Until now 'CO2', 'H2O', and 'ODOUR' are interpreted as occupant sources. These sources are activated only

- ◇ if the name ('CO2', 'H2O' and / or 'ODOUR') has been given in the second data block;
- ◇ if occupants are active in the zone, i.e. the occupant activity level factor and the occupant number defined in `&-SCH-OCC` must be larger than zero;

Description:

Parameters	Description	Default value	Input format, range
No	Occupant type number	-	Integer 1 . . . 5
Sex	Keyword for sex S: MALE MIX FEM:ale	MIX	Keyword string
Age	Age A	20	Real > 0.0
Height	Height H	f(S,A)	Real > 0.0
Mass	Body mass M of occupant	f(S,A)	Real >0.0
Metabolic Rate	Metabolic Rate MR	f(S,A,H,M)	Real > 0.0
Cigarets	Average number of cigarets the occupant type smokes per hour	0.0	Real ≥ 0.0
Name	Occupant type name. This name is only used for identification in outputs of default values and precalculated values. It may not be used as reference in the input file. Reference in the inputfile is always the occupant type number.		
Pol Name	First unambiguous part of the name of a pollutant defined under `&-POL-DES`cription; valid Pol Name's here are 'CO2', 'H2O' and 'ODOUR'	' '	String 10 char
Source Strength	Nominal source strength of the occupant type; if 'd[efault]' is given here in the second data block, the source strength is calculated from `Metabolic Rate` in data block 1;	f(MR) [kg/s]	Real ≥ 0.0

- ◇ if the name of a pollutant defined under `&-POL-DES` matches a valid abbreviation ('CO2', 'H2O' and / or 'ODOUR').
- ◇ if pollutant calculation is enabled under `&-PR-SIMU`
- ◇ if pollutant calculation is enabled under `&-NET-ZP`

There are several possibilities to get the nominal source strength for pollutants 'CO2', 'H2O', and 'ODOUR' of an occupant type. Basically, occupant source strengths are either explicitly input in the second data block, or they are calculated from the metabolic rate. In detail, this means:

- ◇ explicit input in second data block,
  then inputs for `Age`, `Height`, `Mass`, and `Metabolic Rate` in the first data block are ignored;
- ◇ input of the `Metabolic Rate` in column 6 of data block 1,
  then `Sex`, `Age`, `Height`, and `Mass` input are ignored,
  the nominal source strength is calculated from the `Metabolic Rate` if no value or `d[efault]` is given for the `Source strength` in data block 2;
- ◇ input of `Sex`, `Age`, `Height`, and `Mass`,
  if no `Metabolic Rate` is given, it is calculated from these four values,
  the nominal source strength is calculated from the `Metabolic Rate` if no value or `d[efault]` is given for the `Source strength` in data block 2;
- ◇ input of `Sex`, `Age`, and `Height`,
  if no `Mass` is given, it is calculated from `Sex`, and `Age`,
  if no `Metabolic Rate` is given, it is calculated from these four values,
  the nominal source strength is calculated from the `Metabolic Rate` if no value or `d[efault]` is given for the `Source strength` in data block 2;
- ◇ input of `Sex`, and `Age`,
  if no height is given, it is calculated from `Sex`, and `Age`,
  if no mass is given, it is calculated from `Sex`, and `Age`,
  if no metabolic rate is given, it is calculated from these four values,
  the nominal source strength is calculated from the `Metabolic Rate` if no value or `d[efault]` is given for the `Source strength` in data block 2;

That means: sex and age are sufficient input to determine the default value of the nominal source strength for 'CO2' and 'H2O'. The only use of data sex, age, height and mass is to calculate the default of the nominal source strength. It gives reasonable values also for children. The default is replaced by the explicit input of a nominal source strength in data block 2.

The actual occupant source strength for 'CO2' and 'H2O' in a zone is the product of the number and the activity level factor of that occupnat type defined in `&-SCH-OCC` and the nominal source strength defined in, or calculated from, data under `&-OCCUPAN`t.

Cigarettes are regarded solely as source for the pollutant 'ODOUR'. The actual occupant source strength for 'ODOUR' in a zone is the sum of odours of human origin plus cigaret odour. The first term is the calculated the same way as for 'CO2' and 'H2O'. The second term is the product of the number of occupants defined in `&-SCH-OCC` and the source strength stemming from the cigarets that this occupant type smokes per hour, input in the first data block under `&-OCCUPAN`t.

1 cigarette/hour is about 1 olf. A metabolic rate of 104 W also leads to an odour source strength of about 1 olf.

NOTE The units 'olf' for source strength and 'decipol' for concentration must be used for the pollutant 'ODOUR'. They must be defined in the `PR-UNITS` section for input and output.

References

[1] Orme, Malcolm, Liddament, Martin W., and Wilson, Andrew: "Numerical Data for Air Infiltration & Natural Ventilation Calculations" AIC-TN-1994, Reprinted and Updated February 1998, Air Infiltration and Ventilation Centre, Bracknell, UK, 1998

[2] "Infiltration and Leakage Paths In Single Family Houses - A Multizone Infiltration Case Study" TN-27, Air Infiltration and Ventilation Centre, Bracknell, UK, 1990

[3] Allard, F. (Editor): "Natural Ventilation in Buildings — A Design Handbook", James & James, London, UK, 1998

[4] ASHRAE 1985 "Handbook of Fundamentals", Chapter 22, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Atlanta, GA, 1985

[5] Catmull E. and Rom R.: "A class of local interpolating splines", in Barnhill, Rad R. Riesenfeld (eds), Computer Aided Geometric Design, Academic Press San Francisco 1974, pp317–326

[6] Cadloni M., Ferrazzini M,: "Natürlicher Luftaustausch durch Kippfenster", Diplomarbeit ETH Zürich, 1997

[7] Dascalaki, E. et al.: "Predicting Single sided Natural Ventilation Rates in Buildings", Solar Energy Vol. 55 No.5 pp 327-341, 1995

[8] Dorer V. and Weber A.: "COMVEN 3.0 Programmer's Guide", EMPA Dübendorf, CH, 1997

[9] Dorer V. and Weber A.: "Output Options for COMIS", IEA-ECB Annex 23, EMPA Dübendorf, CH, March 15, 1995

[10] Feustel, H.E. (Editor): "The COMIS Model", Special Issue of the Journal Energy and Buildings on Air Flow Modeling Issues, Energy and Buildings, Vol. 18, No. 2, 1992

[11] Feustel, H.E. (Editor): Energy and Buildings Special Issue devoted to Multizone Air-Flow Modeling, Energy and Buildings, Vol. 30, No1, 1999

[12] Feustel, H.E.: "Beitrag zur theoretischen Beschreibung der Druck- und Luftmassenstromverteilung in natürlich und maschinell gelüfteten Gebäuden", Fortschritt-Berichte der VDI-Zeitschriften, Nr. 151, VDI-Verlag, Düsseldorf, Germany, 1984

[13] Feustel, H.E. and Dieris J.: "A Survey of Air Flow Models for Multizone Structures", Energy and Buildings, Vol. 18, No2, 1992

[14] Feustel, H.E. and Raynor-Hoosen A. (Editors): "Fundamentals of the Multizone Air Flow Model – COMIS", Technical Note AIVC TN 29, Air Infiltration and Ventilation Centre, Bracknell, UK, 1990

[15] Feustel, H.E., Allard F., Dorer V., Grosso M., Herrlin M., Liu M., Phaff J.C., Utsumi Y., and Yoshino H.: "The COMIS Infiltration Model", in Proceedings, "Building Simulation '89, The International Building Performance Simulation Association", Vancouver, CDN, 1989

[16] Feustel, H.E., Allard F., Dorer V., Grosso M., Herrlin M., L. Mingsheng, Phaff J.C., Utsumi Y., and Yoshino H.: "The COMIS Infiltration Model", in Proceedings, "10th AIVC Conference", Dipoli, Finland, 1989

[17] Feustel, H.E. and V.M. Kendon: "Infiltration Models for Multicellular Structures: A Literature Review", Energy and Buildings, Vol. 8, No. 2, 1985

[18] Fürbringer, J.-M., C.-A. Roulet, R. Borchiellini: "Evaluation of COMIS", Swiss Federal Institute of Technology, Lausanne, CH, 1996

[19] Fürbringer, J.-M., C.-A. Roulet, R. Borchiellini: "Evaluation of COMIS - Appendices", Swiss Federal Institute of Technology, Lausanne, CH, 1996

[20] Herrlin M.K.: Air-Flow Studies in Multizone Buildings. Models and Applications. Bulletin No. 23, Royal Institute of Technology, Stockholm, Sweden, 1992

[21] Jackman, P.J.: “A Study of Natural Ventilation of Tall Office Buildings”, Inst. Heat Vent. Eng., Vol 38, 1970

[22] Liddament, M.W.: “Air Infiltration Calculation Techniques - an Applications Guide”, Air Infiltration and Ventilation Centre, Bracknell, UK, 1986

[23] Lorenzetti, D.M., and Sohn M.D.: “Improving Speed and Robustness of the COMIS Solver”, in Proceedings, “7th International Conference on Air Distribution in Rooms” (Roomvent 2000), Elsevier, Oxford, UK, 2000

[24] Pelletret, R.: Personal communication, CSTB, Sophia Antipolis, France

[25] Phaff, J.C.: “COMIS Sample Runs”, TNO Report 96-BBI-R 1086, TNO Delft, The Netherlands, 1996

[26] Warren, Peter: “Multizone Air Flow Modelling (COMIS)”, IEA-ECB Annex 23, Document TSR-06-2000, ISBN 1 902177 15 5

[27] Wieringa, J.: ”Updating the Davenport roughness classification”, J. of Wind Engineering and Industrial Aerodynamics, 1992, Vol 41 p. 357-368

Appendix

A Examples

The input file (CIF-file) contains all the data that is necessary for a COMIS simulation run.

The examples illustrate the input procedure. In section 4 'Data Sections' the complete description of the input can be found. It is recommended that the examples and the respective input description are read simultaneously. We will now go through the input files step by step and explain the input data — where they come from and what they are used for. The complete input files can be found on the COMIS v3.1 package distribution CD.

A.1 Minimum Input File

This example is restricted to the absolute minimum of input required. The complete input file 'simple.cif' is listed below. It can also be found on the COMIS v3.1 package distribution CD.

```
&-CR       CRACK

  ___________________________________
 |1.| Prefix and Name | Description |
 |__|        (-)      | [-]         |
 |____________________|_____________|
  ____________________________________________________
 |2.|   Cs    |Exp n   | Length  |  Wall Properties  |
 |__|         |        |         |Thickness| U-value |
 | (airl)     |  (-)   |   [m]   |   [m]   | [W/m2/K]|
 |____________|________|_________|_________|_________|
  ____________________________________________________________
 |3.| Filter 1 | Filter 2 | Filter 3 | Filter 4 | Filter 5 |
 |__|   (-)    |   [-]    |   [-]    |   [-]    |   [-]    |
 |_____________|__________|__________|__________|__________|
*CR1 Crack
      0.003   0.660
      0.0

&-NET-ZONes

  ____________________________________________________________________
 |Zone| Name          | Temp |Ref.   |   Volume    | Humid.| Schedule |
 | ID |               |      | Height|   [m3]      |       |  Name    |
 |    |               |      |       |   H/D/W     |       |          |
 | (-)| [-]           |[temp]| [m]   |   [m/m/m]   |[humi] |  [-]     |
 |____|_______________|______|_______|_____________|_______|__________|
  E01  Examp_Zone


&-NET-LINks

  ____________________________________________________________________
 |Link|Type | Zone ID |  Height | Own    | Act.  | 3Dflow | Schedule Name |
 |    |     |_________|_________| Height | Val.  |   or   |   (10char)    |
 |    |     |    |    |    |    | Factor |       | Press. |_______________|
 | ID |Name |From|To  |From|To  |        |       |        | Ref Link Name |
 |(-) |(-)  |(-) |(-) |[m] |[m] |  [-]   | [-]   | [pres] |     [-]       |
 |____|_____|____|____|____|____|________|_______|________|_______________|
  1    CR1   E01  -2Pa
```

The model consists of two nodes (one zone and one fixed pressure) linked with one link of type 'crack'. For one-dimensional cracks, only one way flow is possible. The resulting flow is therefore zero.

If the zone node is replaced by a fixed pressure node, the resulting flow is non-zero if the fixed pressures are not the same. This is shown in 'simpleP.cif' and 'simpleP.cof'.

NOTE The multiple use of fixed pressure nodes with an airflow component in between is a simple and effective method to determine the pressure-flow characteristic of this component.

If the two nodes are coupled by an air flow component allowing for two way flow, a non-zero flow could result if zone temperature and ambient temperature are different. This is shown in 'simpleW.cif' and 'simpleW.cof', where the crack has been replaced by a window component. The closed window is modeled as a crack with vertical component, thus allowing for single-sided ventilation.

The result part of the output COF-file 'simple.cof' is shown in the following:

```
...

THE OUTPUT STARTS HERE!!

=new=airflow=timestep================================================================
           1 iterations with solver no.           6


 ******************************
 Ventilation output            *
 ******************************


+--ZONES---------------  -----Flow-rates-----
| Zone-ID      Pressure      Total  Imbalance
|                    Pa       kg/s       kg/s
+----------------------  --------------------
  E01               -2.         0.         0.

+--LINKS-----------------  --From---  ---To----  T-link   Dp-link    From->To   To->From
| # Link-ID    Type        Type:Name  Type:Name       C        Pa        kg/s       kg/s
+------------------------  ---------  ---------  ------  --------  ----------------------
  1 1          CR1         zn:E01     sp:-2Pa      20.     0.E+00*          0          0

100.0% (=1) of the 1 links have laminar flow, indicated by Dp*

=end=of=simulation====================================================================
 ----------------------------------------------------------------
 COMIS finished.  (One timestep)
 There are 4 messages, marked ***CER*** in your CER & COF files.
```

In the input file, no start and stop time has been specified. COMIS performs just one simulation time step for the default start and stop time, which are equal.

The format of the output in the COF-file is the same as if the keyword `VENT` had been used under `&-PR-SIMU`. The information given is the following

- ◇ At the beginning of each time step (here we have only one), the number of iterations and the solver used are given:
  `1 iterations with solver no.  6`
- ◇ Under header 'ventilation output', information about each zone and each link follows
- ◇ Zone output

- “Zone-ID: E01” output for zone E01.
- “Pressure (Pa): -2” the pressure at the zone reference plane.
- “Flow-rates Total (kg/s): 0.” the total flow through the zone; in fact it is half of the sum of the absolute values of all flows to and from this zone.
- “Flow-rates Imbalance (kg/s): 0.” the error in the flow balance for this zone; for an ideal solution this error would be zero; the value must meet the tolerances given under `&PR-CONT`rol.

◇ Link output

- “#: 1” the sequence number of the link.
- “Link-ID: 1” the name of the link in the input file.
- “Type: CR1” the name of the air flow component used for this link.
- “From Type:Name: zn:E01” the names of the nodes which are connected by this link; these may be zones, external nodes or fixed pressures, preceded by “zn”, “sp” for fixed pressure or “ex” for (external) facade element number, respectively.
- “To Type:Name: zn:E01” the names of the nodes which are connected by this link; these may be zones, external nodes or fixed pressures, preceded by “zn”, “sp” for fixed pressure or “ex” for (external) facade element number, respectively.
- “T-link (C): 20.” the average temperature of the air flowing through the link
- “Dp-link (Pa): 0.E+00” the pressure difference across the link.
- “From− >To (kg/s)” is the outgoing flow rate.
- “To− >From (kg/s)” is the incoming flow rate; both values being non-zero indicates a two-way flow at large openings.

At the end of the file, the number of timesteps (scheduled events for which the network has been solved) and the number of messages (errors, warnings and notes) is output.

The output produced using the more detailed `2VENT` option is shown in ‘simple2.cof’ (produced by ‘simple2.cif’). The information given is the following

◇ “METEO”: Wind velocity at meteo station (Vmeteo) and the local wind velocity at the building (Vroofheight), followed by wind direction, ambient temperature and humidity.

◇ “LINKS”:

- the velocity in the net area of the openings is given; for industrial ventilation these are the flows that can be measured in the ventilation openings, which enables a direct fit of the model.

◇ “ZONES”:

- the total flow through the zone, the infiltration flow which enters from outside and the ventilation flowrate which enters from other zones.

◇ “Total ...”: summary for the whole building.

◇ “ZONE-GROUPS”:

- grouped output for all zones for which the first two characters of the zone ID are the same; an efficient choice of zone-ID’s allows to get the totals per department or for instance per floor or module of the building.

The error messages can also be found in the file COMIS.CER after the run (messages for ‘simple.cif’):

```
-----------------------------------------------------------------
***CER*** WARNING ***
&-PR-SIMUlation section is missing!
-----------------------------------------------------------------
***CER*** WARNING ***
&-PR-SIMUlation and &-SCH-METeo section are missing!
You may want to add one or both to the input
-----------------------------------------------------------------
***CER*** WARNING ***
&-PR-SIMU: Start time not found
Default value: "1900jan01_" used
-----------------------------------------------------------------
***CER*** WARNING ***
&-PR-SIMU: Stop time not found
Default value: "1900jan01_" used
-----------------------------------------------------------------
COMIS finished.  (One timestep)
There are 4 messages, marked ***CER*** in your CER & COF files.
```

A.2 Factory

This example illustrates step by step

- ◇ the input of a network,
- ◇ the use of large vertical openings,
- ◇ the use of schedules,
- ◇ the use of zone-related pollutants,
- ◇ the use of occupant-related pollutants
- ◇ the use of histogram output.

The complete input files 'factory_1.cif' to 'factory_5.cif' can be found on the COMIS v3.1 package distribution CD.

Building

A factory building consists of two spaces (see figure A.1):

- ◇ a hall with mechanical extract ventilation for combined abrasive and chemical treatment and subsequent bonding,
- ◇ and a naturally ventilated assembly hall.

VOC's are released during chemical cleaning of bonding surfaces. The extract system is designed on the basis of the steady-state concentration determined for maximum source strength and considering the mechanical extract ventilation only and no air exchange with the assembly hall. This concentration must be kept below the threshold concentration (TLV), which is set to 300 mg/kg in this example.

Problem

To evaluate cross-contamination of assembly hall by VOC's from the chemical cleaning and bonding process in the treatment hall.

Model

The factory is modelled as a two-zone network with door, horizontally pivoted windows and roof shed windows as air flow elements. The extract fan is modelled as additional air flow element. Wind pressure coefficients data are taken from literature for simple rectangular building shape, surrounded by buildings of equal height.

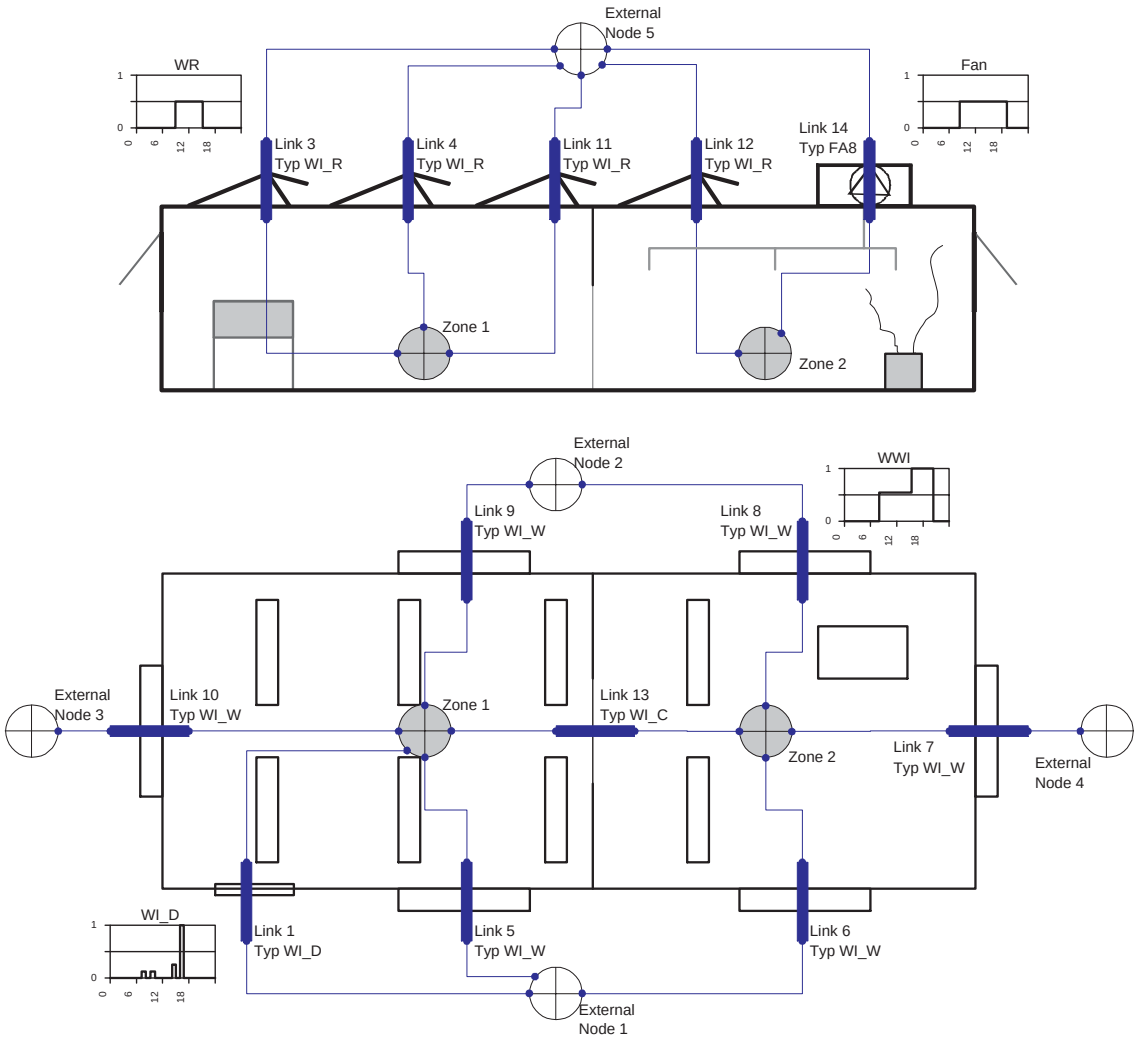


Figure A.1: The factory building with the two halls, the ventilation openings and the mechanical extract system.

Schedules

The simulation is made for one month (720h) with a weather file for moderate climate conditions (April in Zuerich, Switzerland), using a one-hour time step. In the example, the same daily schedule is used for all days (no weekend). Working hours are from 7 am to 6 pm. The fan is running during these working hours. Values for air temperatures are taken from simple thermal modelling estimates. Window opening and contaminant source strength schedules are fixed according to usual working operation, adapted to the one-hour simulation time step. See figures A.2 and A.3.

The occupancy of the zones according to figure A.3 (right) is transferred to the following occupant schedule:

```
00:00 zone_1/zone_2   1.0   0
06:00 zone_1          1.0   2
07:00        zone_2   1.0   5
08:00 zone_1          1.0   7
09:00        zone_2   1.0  10
10:00 zone_1/zone_2   1.0  10
11:00 zone_1/zone_2   1.0   0
```

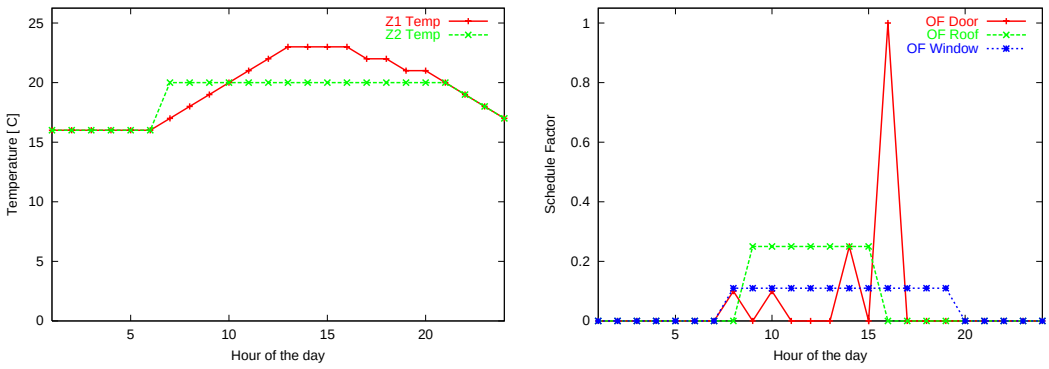


Figure A.2: Schedule values for a 24 h time period for room air temperatures (left) and opening percentage of windows, door and roof sheds (right).

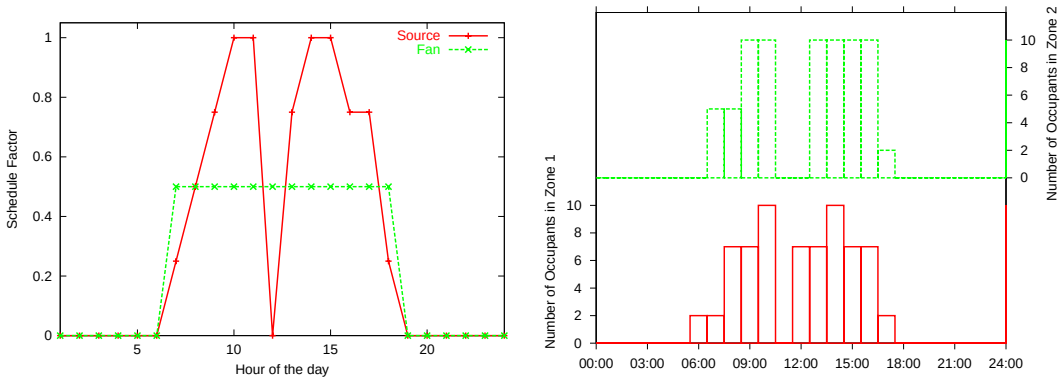


Figure A.3: Schedule values for a 24 h time period for contaminant source strength (nominal value 0.6 kg/h and fan RPM (left) and number of occupants (right).

```
12:00 zone_1          1.0   7
13:00        zone_2   1.0  10
14:00 zone_1          1.0  10
15:00 zone_1          1.0   7
16:00 zone_1          1.0   7
17:00 zone_1/zone_2   1.0   2
18:00 zone_1/zone_2   1.0   0
```

The COMIS histogram

In the CIF-file, output units have been set, the histogram output for both the assembly (zone_1) and the bonding hall (zone_2) have been defined, and two types of histograms have been defined. Here follows a part of the CIF-file:

```
&-PR-UNITS
OUTPUT
conc  mg/kg
conc  ppm


&-PR-OUTP
C1-H1 zone_1 zone_2 OCC1
C2-H2 zone_1 zone_2 OCC1
```

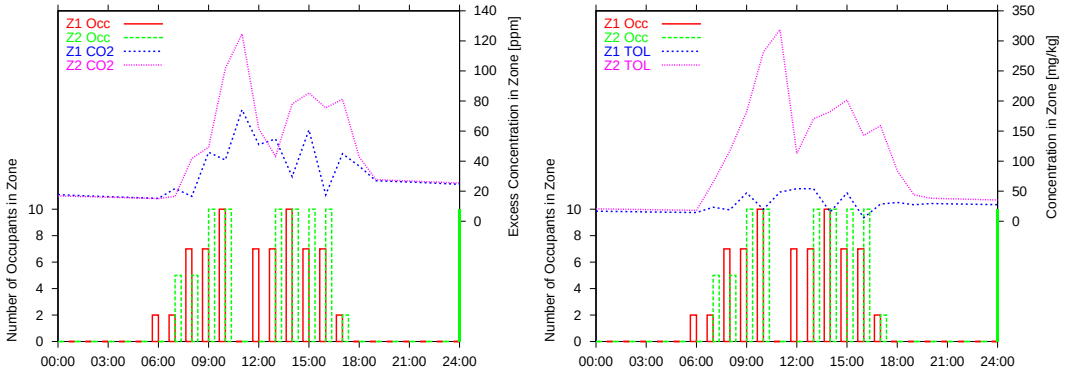


Figure A.4: Occupant Pollutant Load during one day: number of occupants and concentration in zones (left: CO2; right: Toluol).

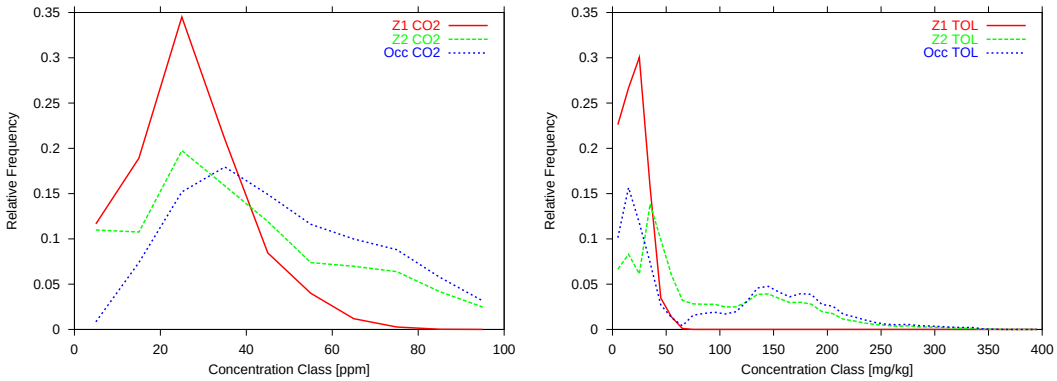


Figure A.5: Occupant Pollutant Load: frequency distribution for zones and for occupants (left: CO2; right: Toluol).

```
&-HISTO
1  40   5.  395.
2  10   5.  95.
```

C1 gives the Toluol concentration, C2 gives the CO2 concentration.

Histogram output for C2-H2 is listed below. The histogram is multiple and like a table. It includes zones zone_1 and zone_2, and occupant type OCC1; each, in their own column.

The occupant-weighted histogram (frequency distribution) counts the concentration of the zones the occupant type happens to be in as many times as there are occupants in. This may change during the simulation as occupants may be scheduled to move into and out of zones.

The types of histograms are defined in the `&-HISTO`gram keyword section. The lowest and highest class values must be given in actual units. I.e. for histogram 1 the unit is mg/kg (as it is used for concentration of pollutant 1), for histogram 2 it is ppm.

In the output, there is one header line describing the histogram type. It is followed by the column titles and a line for the unit of the classes.

NOTE The unit given is that of the classes, not of the frequency! Frequencies are dimensionless values.

Then several rows follow with the center of class in the first column and the frequencies in the second and following columns. `below_range` and `over__range` list the frequencies of values falling outside the histogram range. After that, the maximum frequency and the average is given.

`total_time` gives the total time in seconds for which the histogram has been established. This time may differ slightly from the total simulation time depending on the internal pollution time

step and scheduled events. For occupant-weighted frequency distributions, the total time depends on the number of occupants and the time they spend in the different zones. Therefore it is usually different from the value found for zone statistics, and it has the unit $[pers. \cdot sec.]$

After that, the minimum and the maximum value that has been found is reported.

The last two lines give the conversion factor of the output unit used to the standard COMIS unit.

```
Histogram 2  10 Classes From    5.00E+00 To    9.50E+01 width=    1.00E+01
  ClassCenter      C2-z1       C2-z2       C2-Occ1
                   ppm         ppm         ppm
     5.00000     0.11656     0.10980     0.00846
    15.00000     0.18916     0.10757     0.07388
    25.00000     0.34505     0.19750     0.15184
    35.00000     0.20995     0.15856     0.17941
    45.00000     0.08455     0.11927     0.14917
    55.00000     0.03991     0.07380     0.11617
    65.00000     0.01192     0.06983     0.09994
    75.00000     0.00274     0.06385     0.08813
    85.00000  1.7385E-04     0.04182     0.05768
    95.00000     0.0         0.02479     0.03159
  below_range    0.0         0.0         0.0
  over__range    0.0         0.03324     0.04378
  max_freq       0.34505     0.19750     0.17941
  average       26.35961    41.07990    50.27995
  total_time  2588387.8   2588387.8  15119498.0
  minimum        0.0         0.0         0.45219
  maximum       80.47812   139.67819   139.67819
  Unit_offset    0.0         0.0         0.0
  Unit_mult.  6.5828E+05  6.5828E+05  6.5828E+05
```

Results

Figure A.4 shows the number of occupants and concentration in zones (left: CO2; right: Toluol) for one typical day. Figure A.5 shows the resulting frequency distribution for zones and for occupants (left: CO2; right: Toluol). Occupants reside in both zones, but only during working hours and in varying numbers. Therefore there is a shift to higher concentration values compared with the zones (where low CO2 values in the night are also counted) for the occupant-weighted frequency distribution. For Toluol, we have two peaks. The lower stems from occupants in the assembly hall, which has no source and therefore low concentrations. The higher one stems from occupants working in the bonding hall with the higher concentrations.

The results show that the TLV is seldom reached in the analysed time period.

B Keywords

B.1 Data Section Keywords

&-PR-IDEN tification 48
&-PR-UNITS 49
&-PR-SIMU lation 52
&-PR-OUTP ut 55
&-PR-CONT rol 58
&-HISTO gram 61
&-CR 62
&-NORMCR 64
&-FA 65
&-DS 68
&-F1 71
&-F2 73
&-F3 75
&-F4 77
&-WI 80
&-TD 87
&-RF 89
&-NET-ZON es 92
&-NET-ZL 94
&-NET-ZP 96
&-NET-EXT ernal 98
&-NET-ZT 99
&-NET-LIN ks 100
&-SCH-WIN dow 104
&-SCH-FAN s 105
&-SCH-TEM perature 106
&-SCH-HUM idity 107
&-SCH-SIN k 108
&-SCH-SOU rce 109
&-SCH-OCC upant 110
&-SCH-MET eo Data 112
&-SCH-POL utant 114
&-SCH-MUL ti schedule data 115
&-CP-BUIL ding 117
&-CP-VALU es 118

&-ENV-BUIlding 120
&-ENV-WINd 121
&-POL-DEScription 123
&-POL-FICtive Sources 124
&-OCCUPANt 125

C Set-file COMIS.SET

The COMIS.SET file can be used to describe in which units (e.g., C for temperatures, Pa for pressures, etc.) input values are given and which units COMIS shall use for file output.

Its use for describing unit conversions is strongly discouraged and only remains for backward compatibility with older COMIS.SET files that users may have.

A much better way to define conversion units is to use the `&-PR-UNITS` keyword in the input file itself. If the definition of input and output units is made in the input file only, it is clear which units were used for the simulation (see section 4.1, page 49).

If a set file is to be used for a simulation, it should be copied to the output COF-file by setting the `SET` option in section `&-PR-SIMU` (see page 52).

The COMIS.SET file consists of three parts: The print-level selection part, the part for the input and output units and the part for specification of input and output files. An example for a COMIS.SET file is given in appendix C.4.

Before a COMIS run is made one should assure that the settings in COMIS.SET are correct for the run.

If the command line options -i -o -t -uu and -uf are not given, the respective values are taken from the COMIS.SET file. Entries for these options are found under heading `&-COMIS`, using the keywords `INPUT`, `OUTPUT`, `TABLES`, `USEROUTU` and `USEROUTF` respectively.

C.1 First part

Input of this part is interpreted as follows. It is not allowed to omit any of the first six lines.

The higher the value that is used for lines 3 to 6 the more output is produced.

Line		Range	
1.	write output	$= 0$	in output file (*.COF file)
		$= 1$	on the screen
2.	additional output for	$= 0$	no
	program test runs	$= 1$	yes
3.	print-level for input	$0 \ldots 20$	
4.	print-level for precalculation	$0 \ldots 20$	
5.	print-level for solver	$0 \ldots 20$	
6.	print-level for output	$0 \ldots 20$	
		$= 0$	no output
		≥ 1	Summary at end of simulation
		≥ 2	Every vent timestep:
			Vent2: Building and zone-group summary
		≥ 3	Every vent timestep: Links
		≥ 4	Every vent timestep: Zones
		≥ 5	Every pollutant timestep: Pollutants

C.2 Second part

The first two lines of this part tell how many input and output units respectively follow. Default values are the maximum possible units, 10 and 13 respectively. Values of 00 are allowed, thus enabling to omit unit input in COMIS.SET.

As there is no keyword for the quantities which the units shall be assigned to, they are assigned according to the order of appearance in COMIS.SET. Therefore it is not possible to give units only e.g. for quantities number 1 – 5, and 10, and to omit units for quantities number 6 – 9.

Below the allowed units for both input and output are listed in the sequence from 1 to 13.

The values of the following quantities in the input file are converted by COMIS from input units to COMIS internal units, or from COMIS internal units to output units. The internal units are at the same time the default units for input and output.

NOTE: There are quantities which have only one valid unit. For example, input values for barometric pressure have to be always in kPa. There is also no choice for length input. Depending on the quantity, the unit is always m or mm (see specific headers).

Line	Quantity	Default Unit	Other possibilities
1	Air Leakage	Cm(kg/s@1Pa)	kg/h@1Pa, kg/s@10Pa, kg/h@10Pa, dm3/s@1Pa, m3/s@1Pa, m3/h@1Pa, dm3/s@10Pa, m3/s@10Pa, m3/h@10Pa, ELA4, ELA10
2	Ventilation Mass Flow	kg/s	g/s, g/h, kg/h, m3/s, dm3/s, dm3/h, m3/h
3	Ventilation Pressure	Pa	mmH2O, mmHg, inH2O, inHg, hPa, kPa option -OSR to each unit
4	Temperature	C	K, F, Ra, Re
5	Humidity	g/kg	kg/kg, mass%, %rh
6	Pollutant Source	kg/s	g/s, g/h, kg/h, mg/s, ug/s [a], olf [b]
7	Pollutant Sink	kg/s	g/s, g/h, kg/h, mg/s, ug/s [a], olf [b]
8	Pollutant Concentration	kg/kg	g/kg, mg/kg, ug/kg, m3/kg, dm3/kg, ml/kg, ul/kg, mass%, ppm, ppb, part (any rational combination of g, kg, mg, ug, m3, dm3, ml, and ul are possible, e.g. mg/g, dm3/mg) [a], decipol [b]
9	Component Flow	m3/s	g/s, g/h, kg/s, kg/h, dm3/s, m3/h, dm3/h
10	Wind Speed	m/s	cm/s, kt, ft/min, km/h, mile/h
Output only			
11	Air Change Rate	1/h	1/s
12	Mean Age of Air	h	s
13	Energy	KWh	J

[a] u means μ

[b] units for the special pollutant named `ODOUR`

NOTE The last three units (air change rate, mean age of air and energy) are only used for output unit conversion. The input units must be first, followed by the output units.

NOTE: There is a option '-OSR' which may be appended to the pressure unit. With this option, the outside stack pressure is taken as pressure reference. For further details, see notes on pressure reference on page 33.

C.3 Third part

In the last part of COMIS.SET the keyword for file input and output follows.

In former versions of COMIS, there were several other keywords used for the various parts of the COMIS program. It is still possible to have these keywords in the 3rd part of COMIS.SET. This and also older versions of COMIS simply ignore unknown keywords in this part.

The keywords are followed by the names for the input and output file. If no names are then some default file names are taken. Each keyword starts with a '`&`' sign in the first column.

The only remaining keyword is `&-COMIS`. The syntax is:

```
&-COMIS
```

```
 INPUT <[path]infile.ext>
OUTPUT <[path]outfile.ext>
TABLES "<sep>" name
```

where

INPUT : name for the input file.

OUTPUT: name for the COF-file.

TABLES: Here the project name, which is used for the names of the spreadsheet output files, can be defined. The table created may later be processed with any spreadsheet program. Only the numerical results are printed in this file, separated by the separator strings `"<sep>"` specified. This separator can be any sequence of ASCII characters (including an empty string) and must be enclosed within quotes ("). Examples of separator strings are:

```
"/"       ";"       " "       ""       "*"
```

The names of the spreadsheet output files consist of two parts

COMIS.SET input lines	files created with COMIS
`TABLES ";" yyyyy`	yyyyy-xx.CSO
`TABLES ";"`	-xx.CSO

where:

`xx`	Suffix 2 characters: Depending on what data the input file contains, e.g. PZ for pressures in zones. Which files will be created with COMIS is determined in the CIF-file.
`yyyyy`	Project name 0...5 characters: Defined here under keyword TABLES.

If long filenames are allowed, the project name may be longer than five characters.

The appearance of the keyword `&-COMIS` in the third part of the COMIS.SET file is mandatory.

C.4 Example

There is no output unit for the fan flow specified, as the fan flow is being displayed as a “link” in the units of ventilation mass flow (see Chapter 7 xxx).

```
00 (1=write(CRT.*) on CRT; 0=write(CRT.*) on COF (output file))
00 (print-level for test runs. Range 0,1)
02 (print-level for input. range 0,20)
02 (print-level for precalculations. range 0,20)
03 (print-level for solver. range 0,20)
05 (print-level for output. range 0,20)
00 (number of user units for input)
00 (number of user units for output)
&-COMIS
 INPUT TEST.CIF
OUTPUT COMIS.COF
TABLES "   " EXPL1
```

The required input in the first and second part of COMIS.SET may be followed by explanotory text, which is ignored by COMIS.