# COMIS 3.2

**A multizone air flow calculation program**

# 1.
# COMIS with Simulation Studio
# Tutorial




**EMPA – Materials Science & Technology**

http://www.empa.ch

**CSTB – Centre Scientifique et Technique du Bâtiment**

http://software.cstb.fr

# Scope of this manual

This information manual is not intended to give a complete reference source for COMIS. It rather gives a quick synopsis of the COMIS with Simulation Studio package. The *COMIS with Simulation Studio User Manual* describes the operation of the Simulation Studio front end for COMIS in more detail. The complete input language of COMIS (which is generated automatically by Simulation Studio) as well as a detailed description of the different components of the COMIS standard library are described in the *COMIS User Guide*.
Detailed information about extending COMIS can be found in the *Programmer's Manual*. The *COMIS Fundamentals* (available from AIVC, http://www.aivc.org/ , and not included in the software distribution) contain the physical background information about the COMIS multizone infiltration model.

Information about the latest versions of this software, the availability of paper documents and other related information are available at the following WEB page :

http://software.cstb.fr

or from


CSTB
BP 209
F-06904 Sophia Antipolis
Phone : + 33 4 93 95 67 00
FAX : + 33 4 93 95 67 33

Email : software@cstb.fr


# Acknowledgments


This document is based on the very useful Volume III of the TRNSYS with Simulation Studio software package, *Introduction to TRNSYS for WINDOWS with Simulation Studio*.
Thanks to all the participants of Annex 23 who contributed with remarks and helped me to understand COMIS.
Special thanks to Andreas Weber from EMPA for his continued support as well as to Sabine Taristas for the excellent job on the adaptation of the TRNSYS Simulation Studio to COMIS.



Werner Keilholz
werner@cstb.fr
Sophia Antipolis, August 2005

# TABLE OF CONTENTS


Scope of this manual....2
Acknowledgments....2
Introduction....4
Getting started....5
Installation....5
Starting the software....5
Using Simulation Studio ....7
Beginning the simulation process....7
Modeling the problem in COMIS....7
Entering the problem with Simulation Studio....8
Simulation Studio Assembly Panel Tools....11
Specifying the Parameters....14
Specifying Facades....16
Setting the Global Parameters....17
....19
Running COMIS Simulations....21
Placing Components onto the Assembly Panel....22
Connecting the Components Together....24
Running More Examples....27
Using Schedules....27
Other Simulation Studio Tasks....29

# Introduction

The new, graphical user interface for COMIS described in this manual is based on CSTB's *Simulation Studio* technology, successor of the *IISiBat* simulation environment.

The simulation studio has evolved to house both COMIS and the TRNSYS simulation programs, and includes mecanisms to couple both programs, allowing for coupled air flow and thermal building simulations.

This manual describes how to use the Simulations Studio's graphical user interface to create, execute and manage simulation projects with COMIS.

The Simulation Studio's graphical simulation environment exists also for the TRNSYS simulation software. The TRNSYS version 16.1 and later includes project editors for TRNSYS projects, COMIS projects, as well as coupled TRNSYS-COMIS (thermal-air flow) simulation projects. As a consequence, certain features of the graphical environment may not apply to this COMIS stand-alone version of the tool.

# Getting started

## Installation

If COMIS with Simulation Studio has not been installed on your computer yet, simply introduce the COMIS 3.2 CD-ROM, start the setup program and follow the instructions on the screen. We recommend to install the software into the main directory of your hard disc (such as \comis32), and not under Program Files.

Note for TRNSYS users:

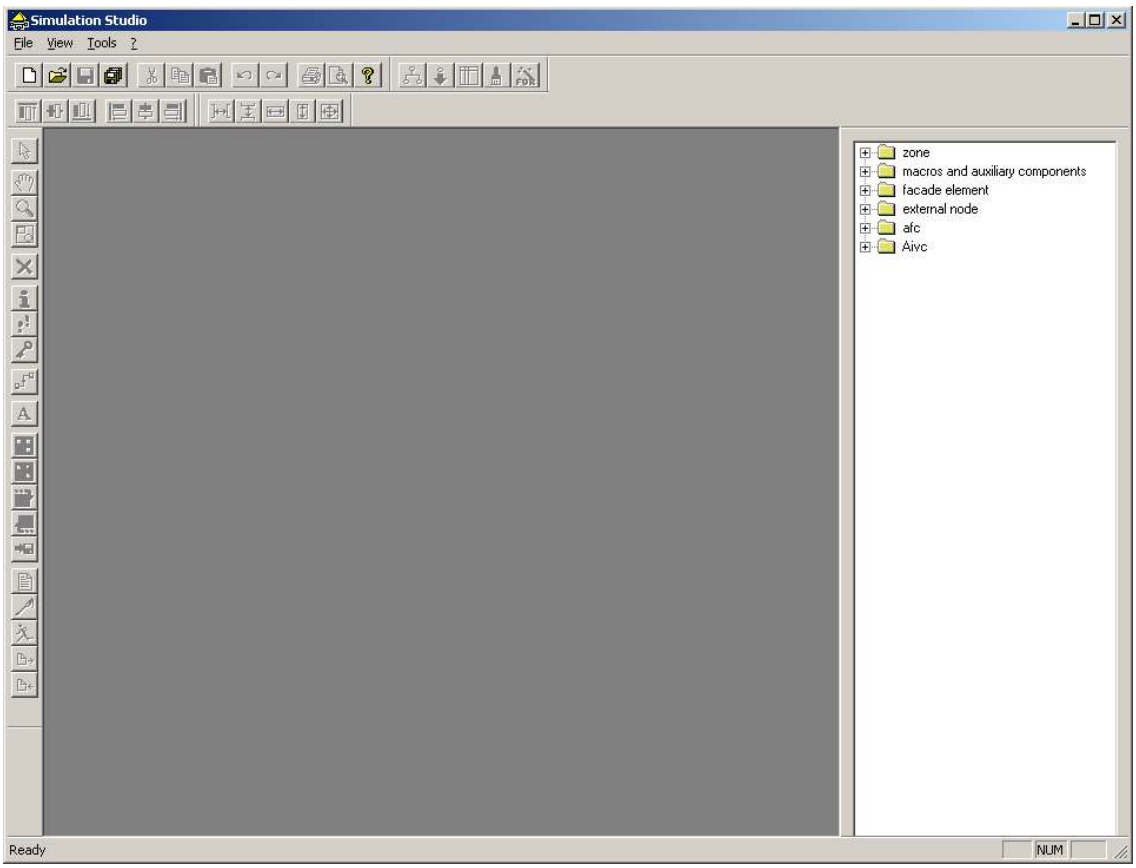
If you plan to use COMIS with TRNSYS 16.1 or later, you can create the link between the two environments by adding the following lines to the Studio.ini file in the . \Trnsys16\Studio\exe directory :

```
[COMIS]
COMIS_EXE=\comis32\binary\comis32.exe
```

You can then work directly in the TRNSYS 16.1 environment to execute COMIS simulations, TRNSYS simulations, as well as coupled simulation projects.

## Starting the software

The installation procedure creates a *COMIS 3.2 Simulation Studio* icon in the WINDOWS environment. Use this icon to start COMIS.



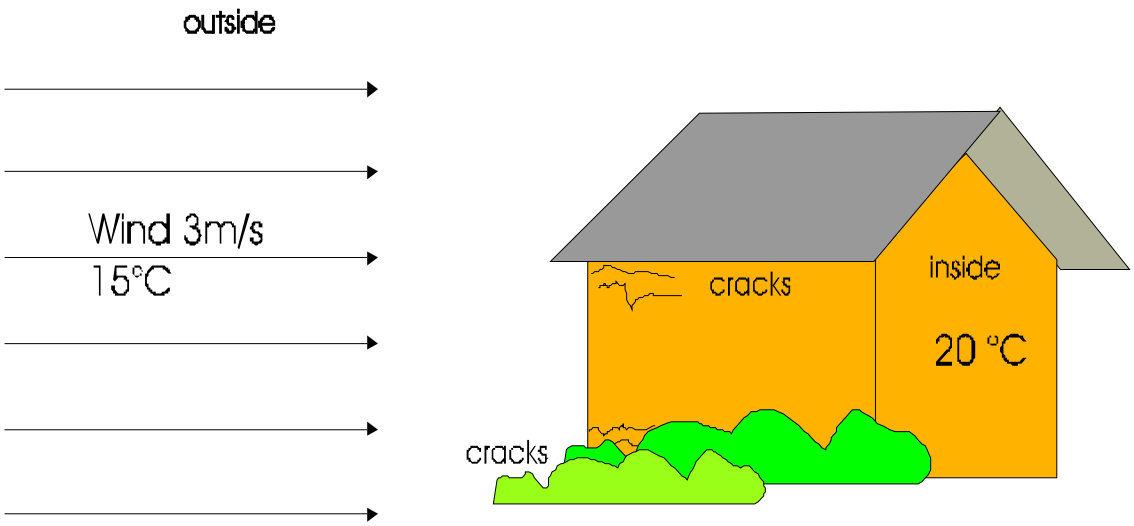

*Simulation Studio main window*

# Using Simulation Studio

## Beginning the simulation process

This section introduces a first, simple example of a air flow simulation with COMIS, using the Simulation Studio interface.

To familiarize users with the process of going from idea to result, a simple example will be shown in every stage of the simulation process. For the purpose of this tutorial, we will consider a single-zone structure which has only four connections to the outside – the North, East, West and South facades (imagine a block hut with a tightly closed door only (no windows, no leaks in the roof). We assume a steady wind at 3m/s blowing from west to east.



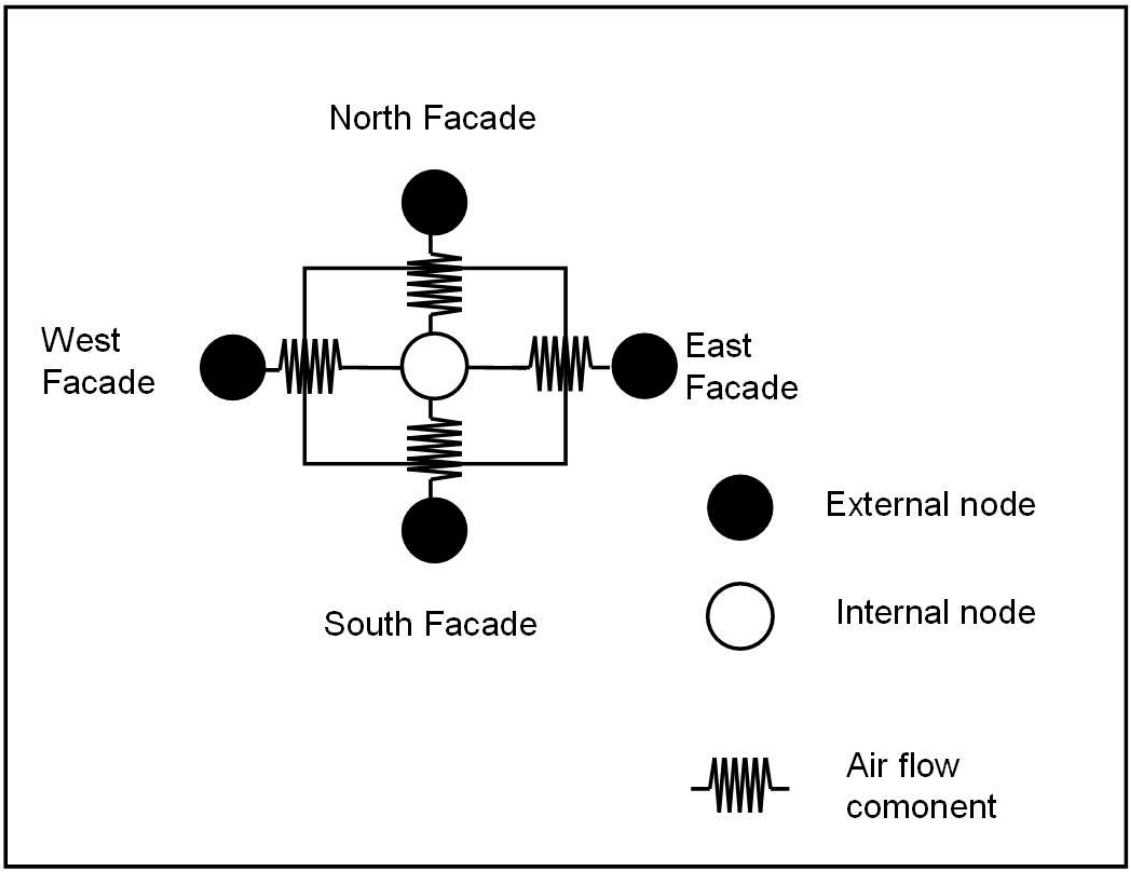

*The problem - Uncle Tom's Hut*

### Modeling the problem in COMIS

To describe this problem in COMIS, we have to define a network of *pressure nodes*, connected by *air flow components* (AFCs). Pressure nodes are volumes in the building, in which we assume the air to be well mixed and the pressure stationary. They may represent a room, a group of rooms, a floor or an entire building, depending on the structure / granularity of the problem. Of course the before mentioned assumptions (fixed pressure, well mixed) may not always be satisfied in reality. By defining a certain volume to belong to one node, we simply state that what happens 'inside' this volume is of no interest for us.

In our example, we have only one single room with no separations; thus we only need one zone (the building).

In addition to internal zones, we need to define the air pressure field around the building. This is done by creating *external nodes* and connecting them to the zones.




*The problem modeled in COMIS*

## Entering the problem with Simulation Studio

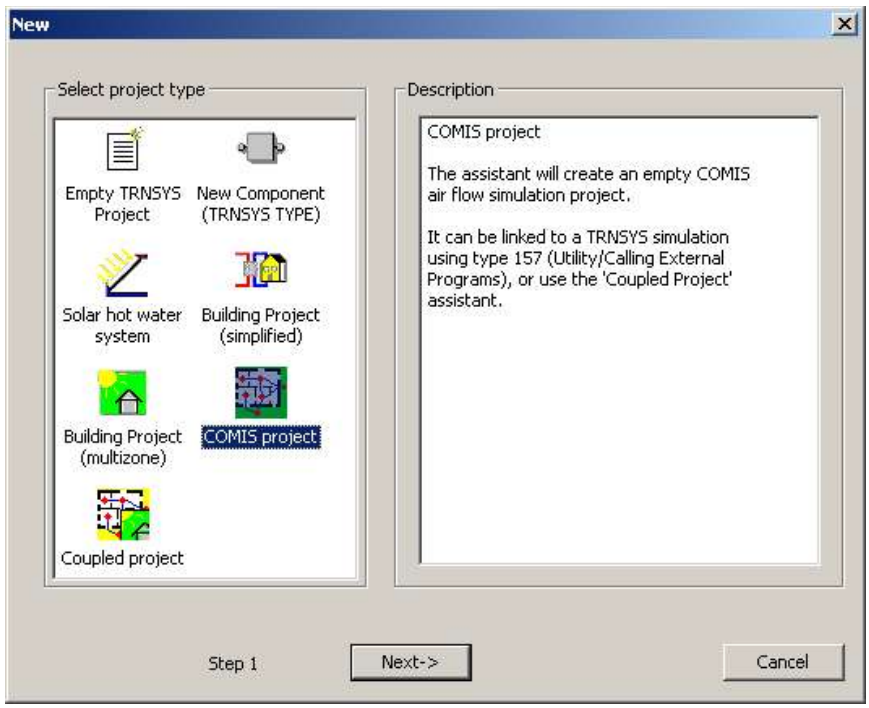
As you can see from the above figure, the problem description is a network of pressure nodes, interconnected by air flow components. This problem description has to be translated into the input language used by COMIS, the COMIS Input File (CIF). The graphical user interface of Simulation Studio allows you to enter this information graphically with the help of the *Assembly Panel*. Simulation Studio then generates a COMIS input file (CIF). Although this is done automatically, the user should still have a notion of the CIF syntax described in the *COMIS User Guide*, in order to be able to track down problems.

The first step in creating such an assembly, is to create a new project document:

➔ select File/New

This opens a window which allows you to select from a list of available project types.

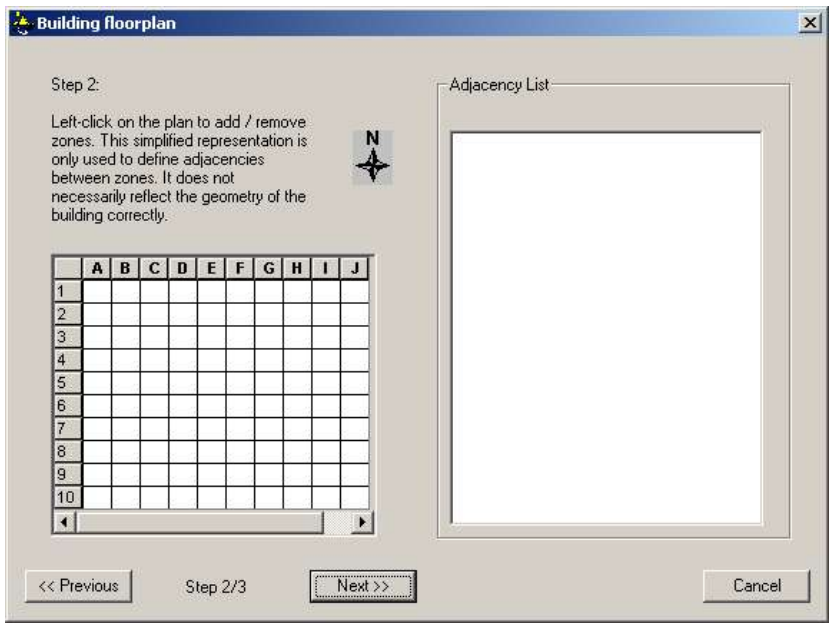
➔ Select “COMIS Project”




The project selection window

➔ Push the Next-> button

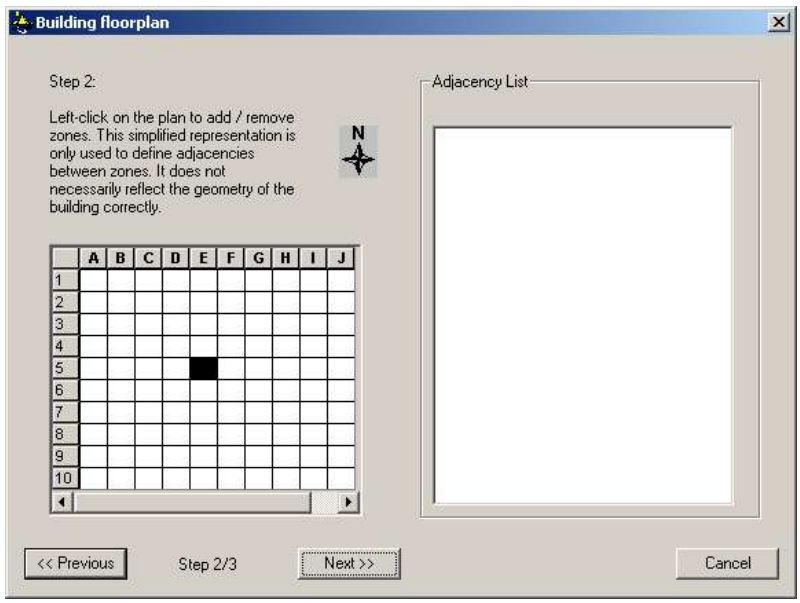
A building floor plan layout editor will open.




The building floor plan editor

In this window, air nodes can be defined by clicking at the grid points on the left side of the window. For our simple example, we will only create one air node.

➔ Click (for example) on the box in line 5, column E

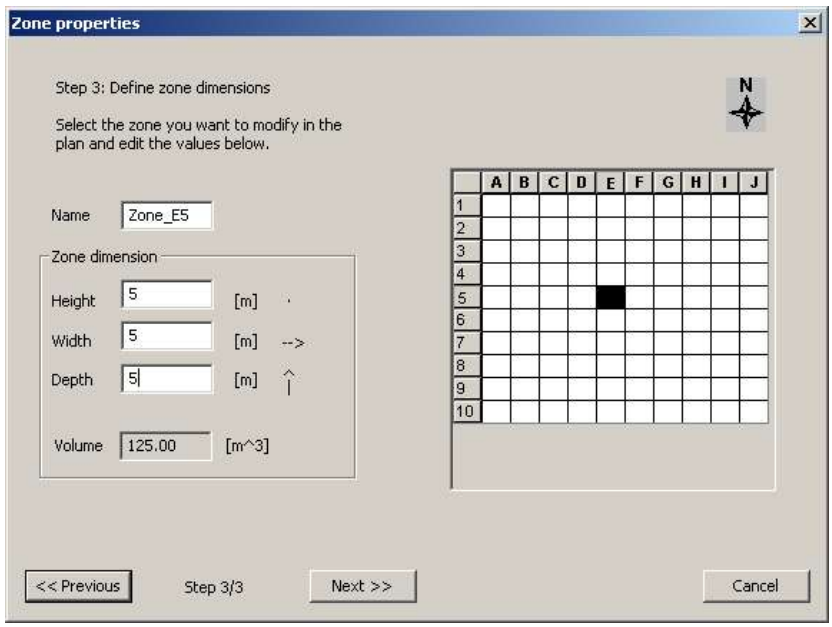



The building floor plan editor with one zone defined

➔ Push the Next-> button

You must now specify the dimensions of each node in the project. To do so,

➔ Click (again) on the box in line 5, column E and fill in the width, height and depth of the node, as "5" respectively.

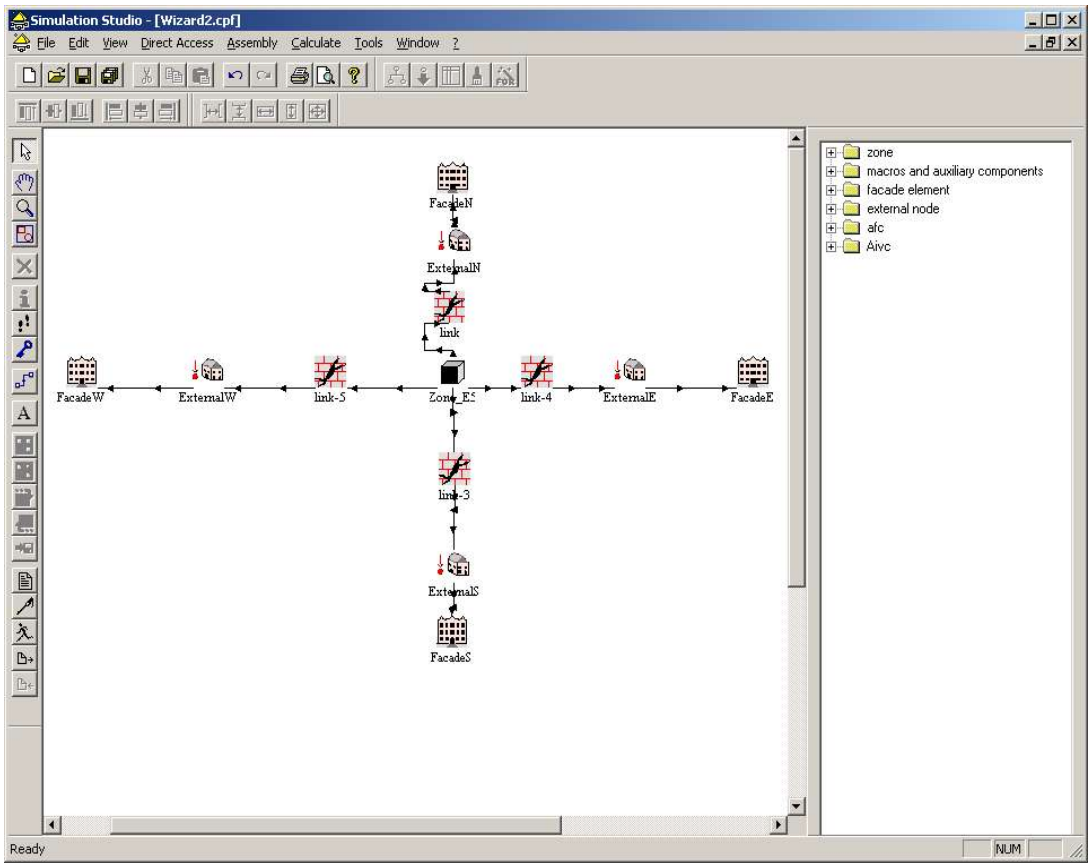



Specification of node volume by clicking on the node and typing the zone dimensions

➔ Push the Next-> button

Now a project template is created. This default project contains the air node (zone) we have defined, as well as 4 façade elements (North, South, East and West).

Note that you may have to scroll or zoom (see below) to see the project completely.



Automatically created project (1 zone)

To create simulations graphically in the Simulation Studio, the users places icons onto the working surface of the Assembly Panel window, creates links between the components, defines the parameters, and runs COMIS.

The project creation wizard has done most of the job for us – a template project is already created. The following section gives a brief summary of tools allowing to manipulate the project.

## Simulation Studio Assembly Panel Tools

A quick review of the most important Simulation Studio Assembly Panel tools is given below. A more detailed description of these tools and of the operation of the Assembly Panel window is given in the *Simulation Studio User's Manual*.



**Select Tool**

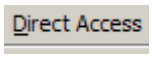
The Select Tool in the Assembly Panel performs many important operations including:

- select component(s) on which actions will be performed
- move components to new locations within the assembly panel

- move the links between components
- view the list of parameters associated with each component
- rename component models

 **Erase Tool**

The **Erase** Tool allows the user to delete a model or a link between two models. Selecting this tool and clicking on a model deletes the model and all of its links to other components. Selecting an object in the assembly panel and clicking on this tool removes the selected object.




**Direct Access Tool**

The **Direct Access** Tool allows users to retrieve standard COMIS models and place them onto the Assembly Panel. It exists as a menu (on top of the main Studio Window) and as a tree view (on the right side of the main window).

 **Link Tool**

The **Link Tool** allows the user to specify possible air flow paths between two component models. The direction of air flow is defined as positive from the first model to the second model. To connect two components, the user must select the **Link** Tool and then click on a first component. The cursor will change to reflect an ongoing link operation. Users must complete the link process by clicking on the component model to which the link will go. A segmented line will be drawn between the two components to indicate information flow between the two components.

 **Write Tool**

The **Write** Tool will create a COMIS input file from the information in the Assembly Panel window, but will not call the COMIS simulation engine to perform the calculations. This input file may then be viewed using the **Editor** Tool.

 **Run Simulation Tool**

The **Run Simulation** Tool allows the user to run the COMIS simulation program on the currently opened project in the Assembly Panel window. Clicking on this tool starts the following process:

- The project is checked for any obvious errors (verified).
- The COMIS input file is created for the project.
- The COMIS simulation program is called with the newly created input file.

 **Input File Tool**

The Input File Tool opens a text editor with the COMIS Input File (CIF) created by the Simulation Studio.



**Output File Tool**

The Output File Tool opens a text editor with the results computed by COMIS.



**Global information**

The **Simulation Card** Tool allows the user to specify the simulation control parameters which are required by the COMIS program. These control cards include the start and stop times of the simulation, the solution method statement, weather data, schedules, etc. Refer to the *Simulation Studio with COMIS User Manual* and the *COMIS User Guide* for more information on the control statements.

## Specifying the Parameters

An important step in the creation of a simulation in Simulation Studio is the specification of the required parameters for each component model. These variable windows can be accessed in the assembly panel by double-clicking on the icons.

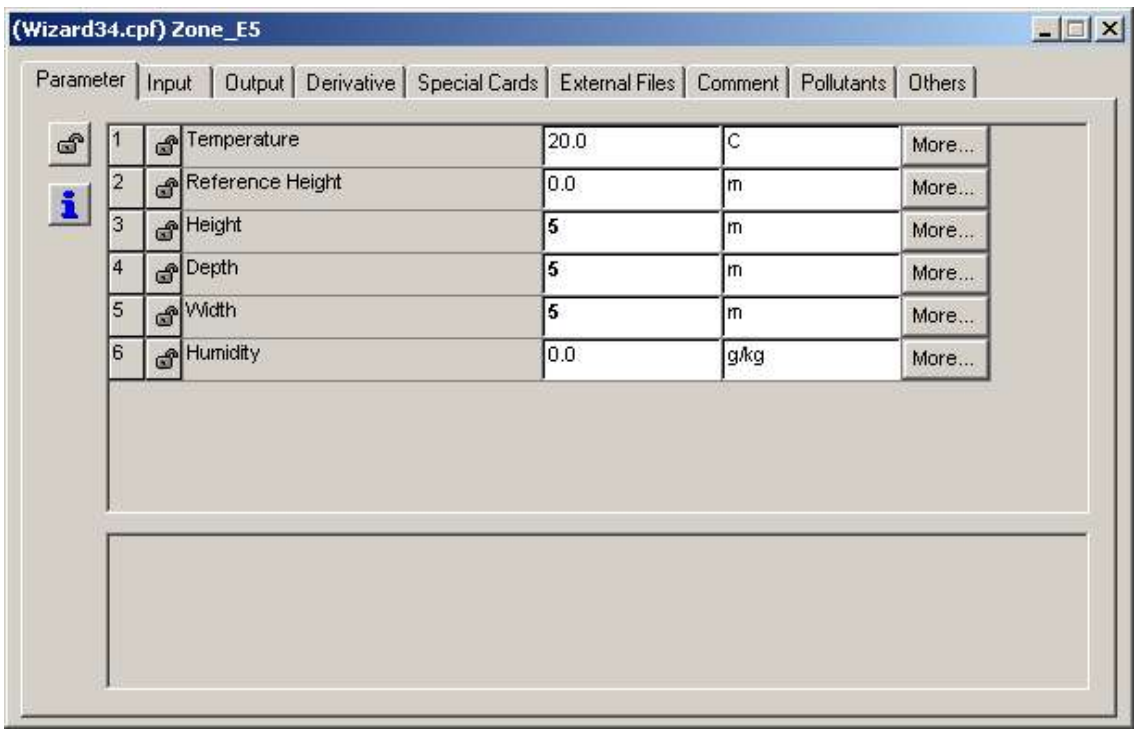
In the case of the zone component, 6 parameters must be specified. For reference, the required statement for the zone in the input file (as described in the *COMIS User Guide*) is:

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

Zone_E5 Zone_E5       20.0   0.0      5/5/5        0.0
```

The *zone ID* and *name* (zone_E5) is the name of the icon in the assembly window (it can be changed by clicking on it). The section '*Schedule Name*' is filled in by Simulation Studio automatically, if schedules are attached (attaching schedules is described below).

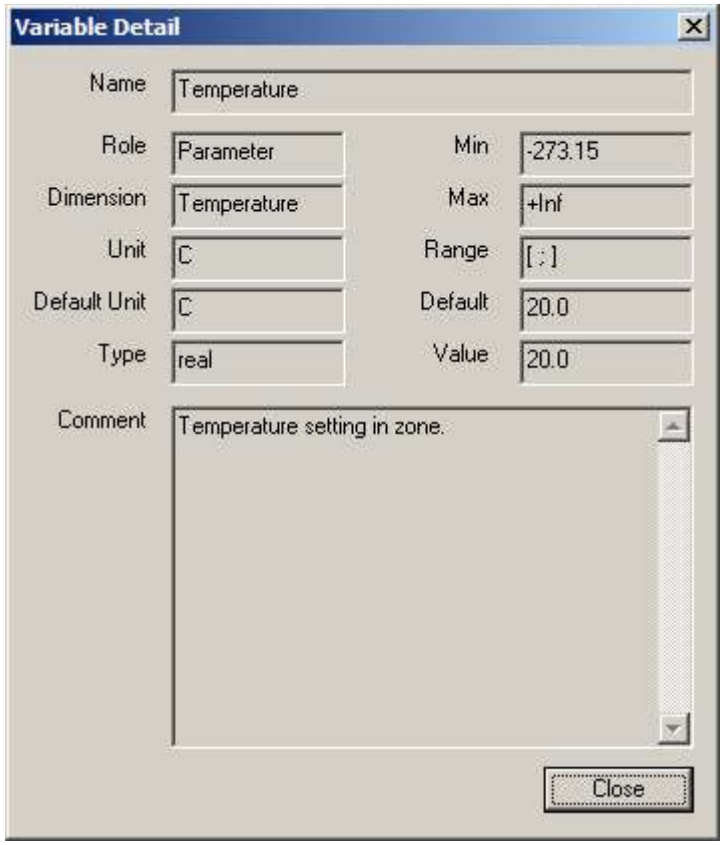
To specify the required parameters for the *zone* component model, click on the Select Tool and then double-click on the zone icon in the project. A window will appear listing the four required parameters for this component as shown below.



*Parameters Window for the zone component model*

Note that default values have been filled in by the project creation wizard. Simulation Studio also 'knows' values for the minimum, maximum, and default values for all of the variables. To demonstrate this feature, attempt to modify the Height (Parameter 3) to -42, for instance. To change the value, click just to the right of the numerical value and use the backspace key to delete the existing value; then simply type in the new value of -42. The value will automatically change back to 5, indicating that the value you typed is not allowed.

To get detailed information about a parameter, the user should click on the 'more' button to the right of the parameter. A variable information window will appear similar to the one shown below. After reviewing the information, close the variable information window for the parameter.




*Variable Information Window for the First Parameter of the zone component*

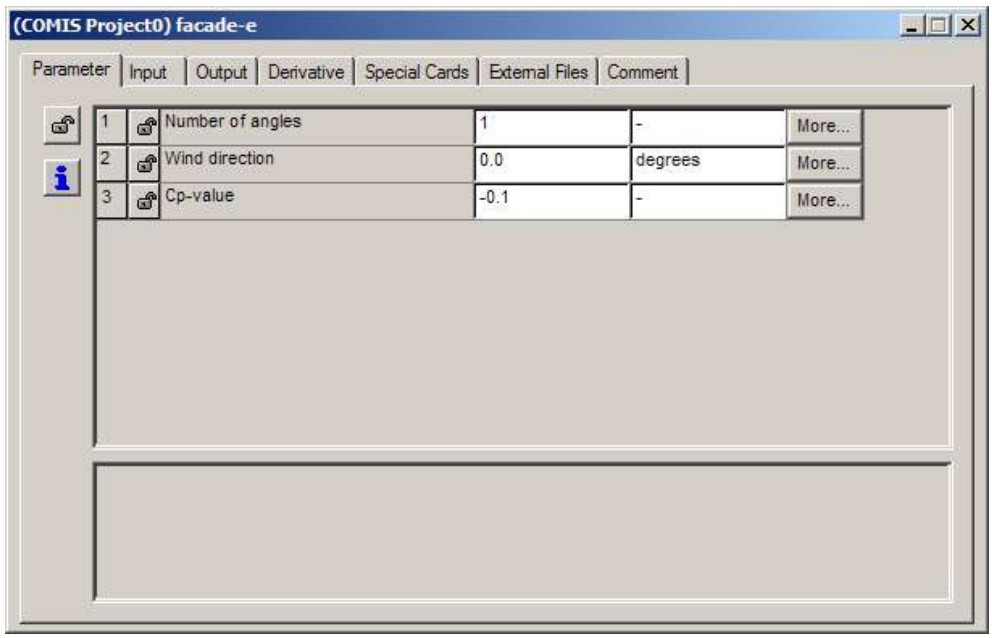
Close the Parameters window using the Close button in the upper right corner (or Alt-F4) and return to the Assembly Panel window. At this point, the zone is completely defined

In a normal Simulation Studio session, the user would now enter the parameters for each of the component models on the Assembly Panel window. However, to save some time in this demonstration, only the facade element component models will be completely specified. For the other components, we will use the default values.

## Specifying Facades

Facade elements are important to 'connect the building to the outside'. They are always linked to an external node. Each facade element is specified by a list of n data pairs (cp-value[1], wind direction). For our example, we will assume that for facade elements only one data pair is known: a cp-value of -0.1 for a wind direction of 0°. To enter these values:

➔ open the parameter window of the facade element 'FacadeW'. The value for '*Number of Angles*' already defaults to 1, so we don't have to modify it. The first data pair defaults to a Cp-value of 0 at an angle of 0°. So we only have to change the value for '*Cp-value*', the last parameter of the component. Click inside the box containing the value 0, delete it and replace it by **-0.1**. Close the window.




Defining Cp-values

This first facade will be used to represent the west facade of our hut.

➔ Repeat the process for the east façade ('FacadeE'), using a cp value of 0.1

[1] See the COMIS Fundamentals for the exact definition of cp-values; be careful not to mix them up with cs-values

**Note:** Specifying only one cp-value for 2 of the 4 facades is not realistic. It is done here for purpose of demonstration only. In particular, changing the wind direction during the simulation will have no influence in this case. In a realistic simulation, cp-values for several wind direction should be specified for all facades. If you do not have access to wind tunnel data, you can use the Cp_calc program (available in the 'files' section of the COMIS discussion group on http://groups.yahoo.com/group/comis ) or TNO's online service at http://cpgen.bouw.tno.nl .

## Setting the Global Parameters

There is one remaining task in creating the simulation: to specify the required global data, such as weather data, occupant description, pollutant description, desired output, etc. This information is component-independent and is entered separately from the components in the Global Information window.

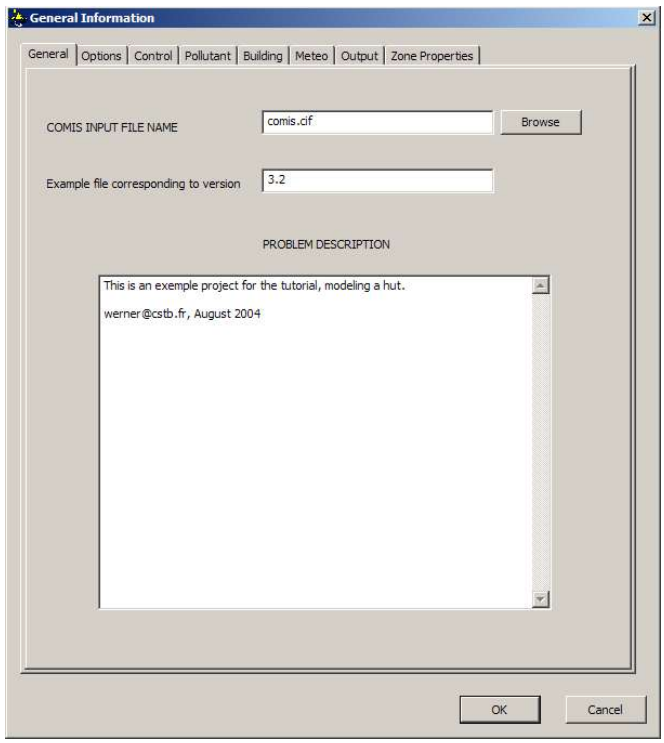
➔ To access this window, click the Global Info Tool .

This action will produce the Global Information window as shown below. The global information window contains a series of tabs, which are accessible from the buttons on the top of this window. The values that are contained in these windows are initially the COMIS defaults.

In general, it is important to verify the following points:

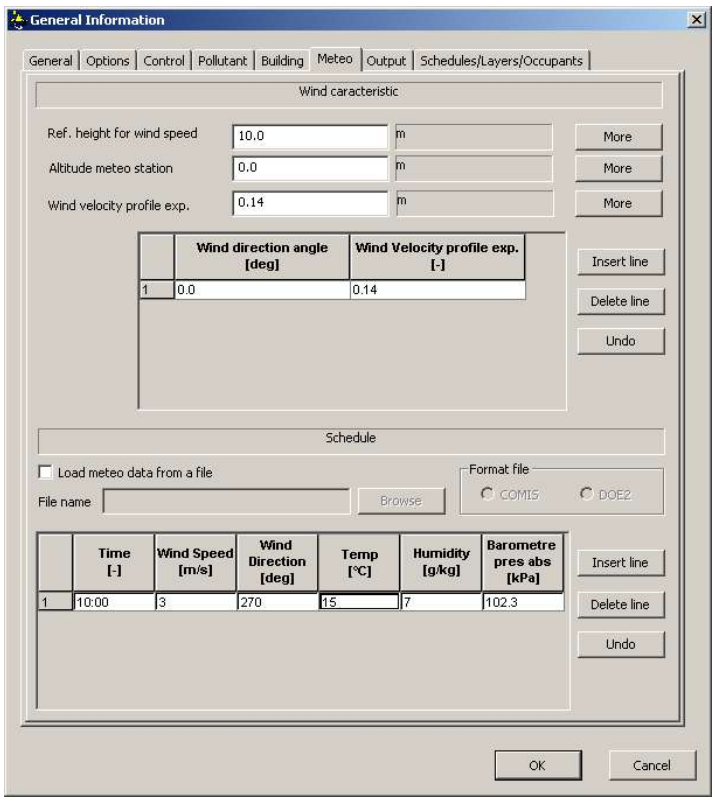
- the simulation start time should be adapted to all schedules used, otherwise scheduled events might not occur within the period simulated. Since our simple example does not use any schedules, we can just use the default values. To modify the start and stop times, click on the "Options" button in the *Global Information Window*. See the COMIS User Guide for allowed date/time formats.

- at least one line of weather data should be given




*Global Information Window*

➔ To specify meteo data, use the *Meteo* tab.



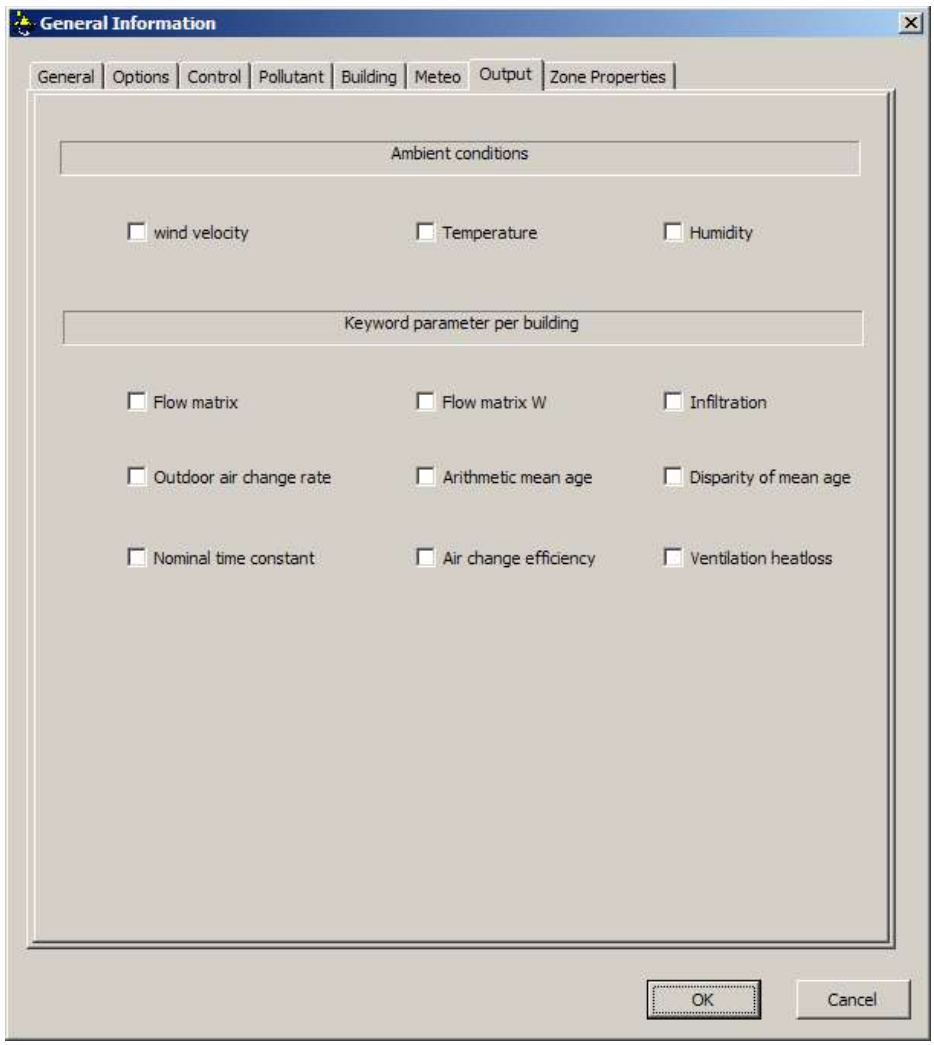

*Meteo tab in the global information Window*

To enter information into any input box in this window, click on the input box with the mouse, delete the existing value with the backspace key, then enter the new information.

➔ Enter the values given in the figure above for our example. 270° indicates wind blowing from west to east (wind angles are measured clockwise from the north).

The remaining global data should be checked; but in our example the default values are sufficient. Please consult the COMIS User's Guide for details.

To define what output should be calculated (and stored in files for later use) the output tab in the Global Information window has to be used. This menu allows the definition of different types of output.



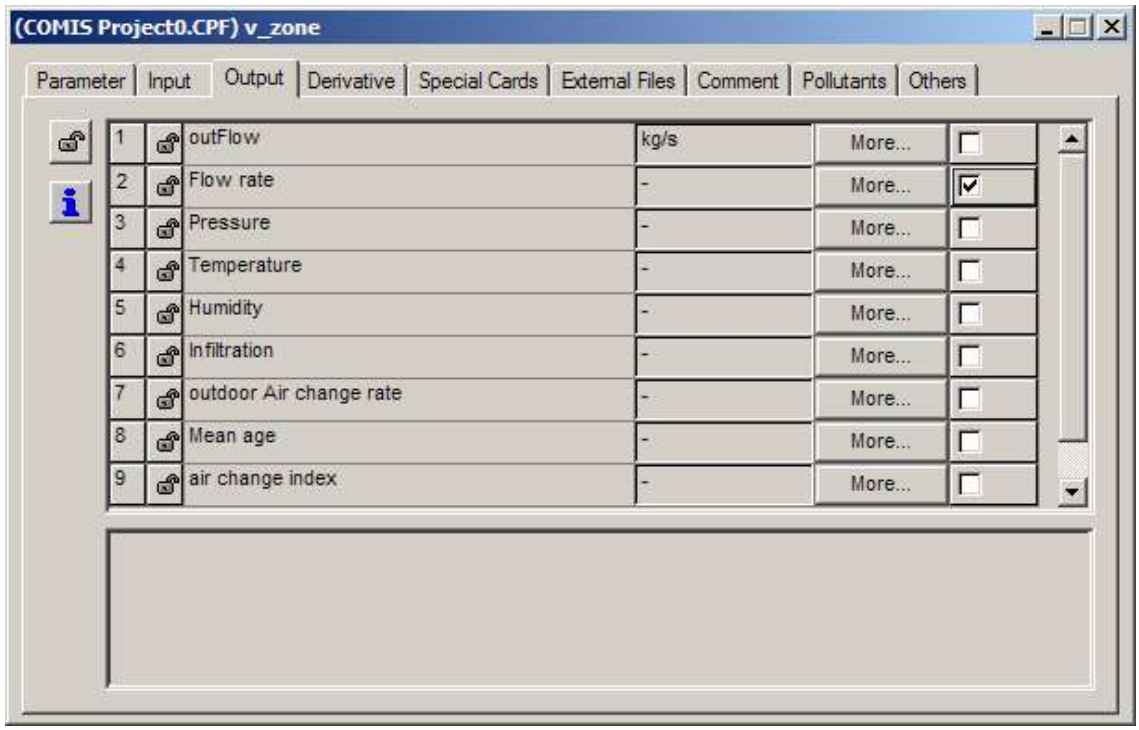

*Global Output Options Window*

➔ At this point, quit the *Meteo* and *Global Information* windows to return to the Assembly Panel window.

Output options are also available for certain components in the assembly panel. For example,

➔ double-click on the zone icon, and go to the Output tab.

By checking the checkboxes in this window, additional output can be produced for this component.




*Selecting component outputs*

In the example given in the figure above, the user is interested in the Flow Rate computed by the zone component. Related data is stored in CSO (COMIS standard output) files. One data line per timestep is produced.

At the latest at this point, it is wise to store your work in a file. The wizard saves the automatically created version in the MyProjects directory, but your modifications would be lost if you do not save the project.

➔ Select File/Save to store your project. Use File/Save As… to store it under another name or in another directory.

Simulation Studio stores projects in .CPF files (COMIS Project Files). These files contain all project information, including icons, while the COMIS Input File (CIF) only contains the numerical values for the simulation.

## Running COMIS Simulations

Running the simulation is actually very simply done by clicking the Run Simulation Tool (). Simulation Studio performs several tasks at this point. First, it checks the project for obvious connection errors.

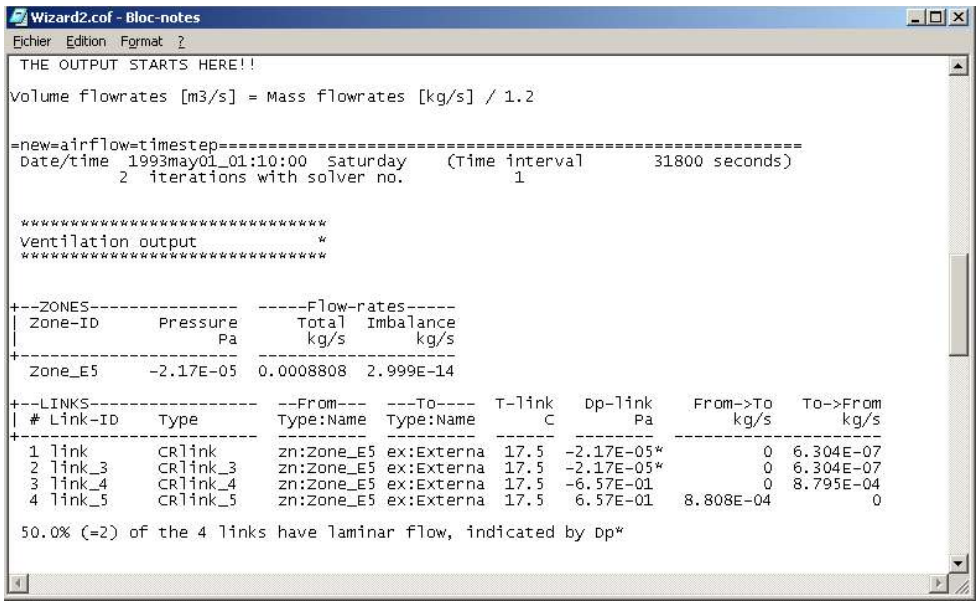
After verifying the project, Simulation Studio generates the COMIS input file which will be run by COMIS. Note that this file is stored in the path for the input file as entered into the Simulation Control Cards window (General tab), and can be different from the project (.CPF) file name.

➔ push the run button ().

**Note:** If you have installed the software in the \Program files directory (or any other directory containing spaces), you may have to adapt the COMIS input file name in the Global information window (). You may for example use \comis.cif, or \myproject\comis.cif (the \myproject folder must exist).

When the calculations are complete, the window in which the COMIS program has been executed may either be removed from the screen automatically or remain visible, depending on the operating system configuration. If the window remains visible after the simulation is finished, you can close it with the windows-closer.

A text editor containing the COMIS output file should open automatically:




The user should immediately check the COMIS output file after a simulation run. The COMIS Output File lists warnings or errors detected in the input file and runtime errors that have occurred during the simulation.

In the case of the example created with the assistant (wizard), the output file has a default name like *Wizard1.COF (Comis Output File* - the name of the output file is derived from the CIF name given in the Global Information window, where '*.CIF*' is replaced by '*.COF*'. It is located in the same directory path as the input file).

The output file is the first place to check for warnings or errors in the COMIS simulation.

➔ To access the output file at any time, click on the Editor Tool () in the Assembly Panel window.

The COF file lists basic information computed by COMIS. Note that from/to conventions use the orientations of the links in the project window. In the example shown above, a flow of approx. 8.8e-4 kg/s through link_4 is computed in the To->From direction. This means, that the actual flow is in the opposite direction of the arrows in the project, i.e. from the East façade (FacadeE) to ZoneE5. (Due to the poor Cp value data we have used, this direction will not change if the wind direction is modified; see also the exercises at the end of this tutorial).

In addition to the .COF file, so-called .CSO (COMIS Standard Output files) are produced for component-related output. The first line of each CSO file contains the physical unit used and the COMIS output option which produced the file. The second line contains numbers of the elements which are stored in the respective columns.

## Placing Components onto the Assembly Panel

In our first example, we have used the Studio's assistant to create the project. It is of course possible to freely modify the created project, by erasing and adding components and/or links.

To erase components, one simply selects them by clicking on the component's icon. Then the component can be deleted by using one of the following methods:

- hitting the delete (DEL) key
- choosing Edit/delete from the main menu
- choosing Edit/cut from the main menu

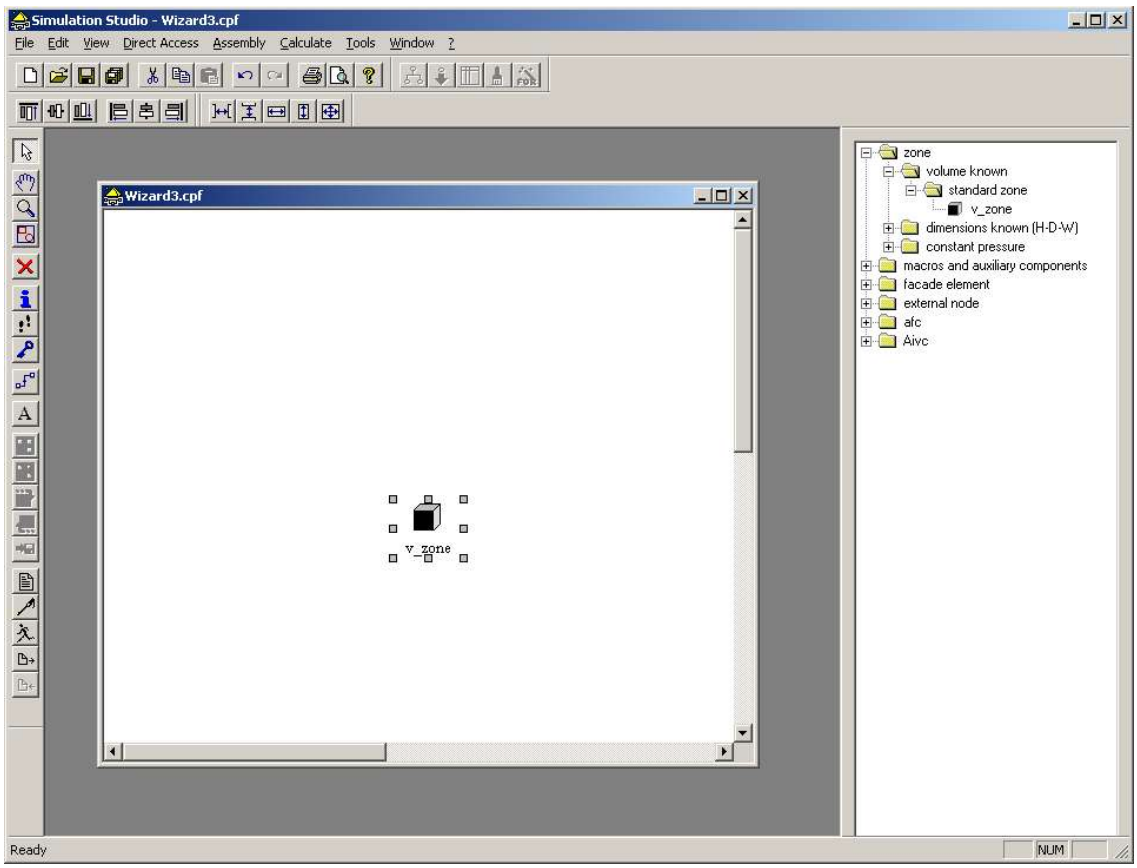
Note that it is also possible to select several components by dragging a rectangle around them with the mouse.

➔ After saving your project (File/Save, File/Save As...), select all components and delete them. (Alternatively, select the components one by one and delete them).

The **Direct Access** Tool is the quickest and easiest method for placing components onto the Assembly Panel.

The first component necessary for the example system we are using is the *zone* component, representing the only room of our 'building'. To place this component onto the Assembly Panel, follow these steps:

- ➔ Click with the left mouse button on the '+' next to the word in the tree view on the right side of the window, or double-click on the word 'zone'. This opens the library.
- ➔ Open the 'Volume known' sub-library. This selects a zone model which is specified by its volume, rather than its dimensions. Select 'Volume known'.
- ➔ The mouse pointer will turn into a crosshair indicating that you have selected a component and need to place it on the Assembly Window. Select an appropriate place on the assembly window and press the left mouse button again. The Simulation Studio icon for the *zone* component will appear on the Assembly Panel similar to that shown in the figure below.




*Zone component Placed On the Assembly Panel Window*

and waiting to be connected to other components in the system. Now, we need to drag the other components onto the Assembly Panel and define their variables. The remaining components and the path used for the Direct Access tool are listed below:

- ➔ Facade Element:

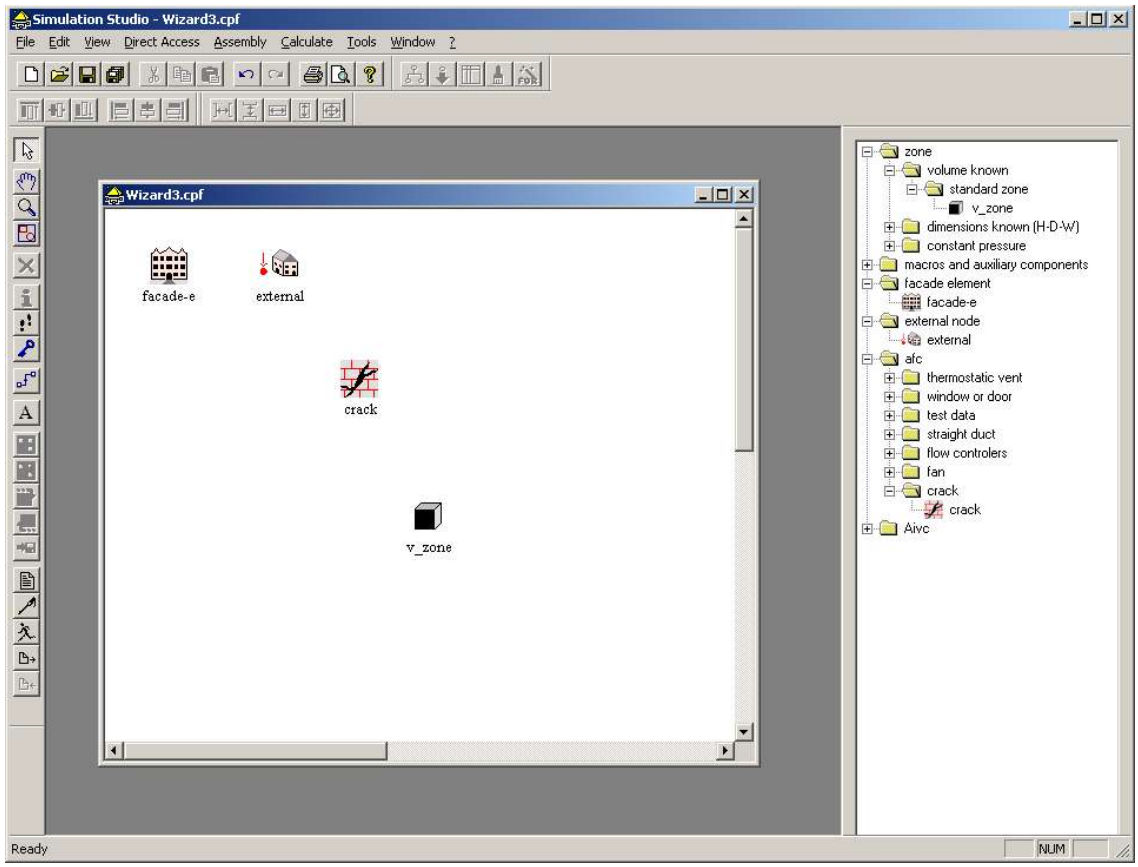
*COMIS / facade element / **facade-e***

- ➔ External Node:

*COMIS /external node / **external***

➔ Crack:
*COMIS / AFC / crack / **crack***

When all the component models have been retrieved using the Direct Access Tool, re-arrange the models on the Assembly Panel (using the Select Tool and dragging the icons to their new locations) so that they resemble the figure below. The precise placement of the components is not extremely important. It will often become apparent, as the links between the components are created, what the logical placement will be. The components can always be moved with the Select Tool.




*Assembly Panel with Necessary Example Components in Place*

Since we need exactly the same component for the floor, we will simply copy it :

- ➔ Select the façade element
- ➔ select Edit/Copy (or hit Ctrl-C)
- ➔ select Edit/paste (or hit Ctrl-V)

## Connecting the Components Together

The next step in the development of this example is to define how the components are connected together. The connections between components show in which way air might flow in the network. For example, a zone ‘kitchen’ may be linked to a zone ‘living’ by a ‘door’, i.e. there is a door between the kitchen and the living room. These connections

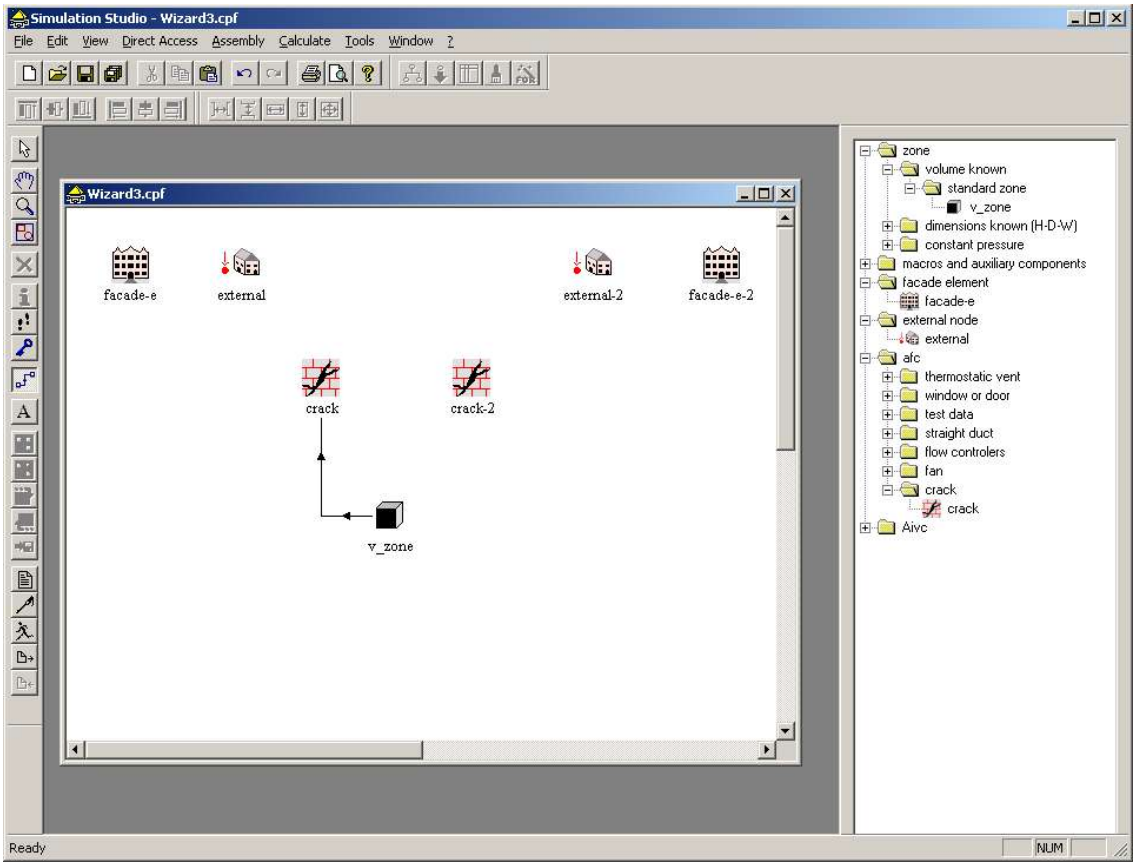
actually define lines of code within the `&-NET-Links` section of the COMIS Input File for each connection that links two zones.

In our simple example, no 'internal' air flows occur, since there is only one zone. So we only need to connect the zone to the external node. In COMIS, pressure nodes can never be connected directly. An Air Flow Component (AFC) has to be kept between them.

We have chosen to connect our zone to the outside with the help of *crack* components. The crack is a very popular, multi-purpose AFC that can be used to model just about any opening. It is perfect for a floor and ceiling with cracks.

To connect the zone to the crack, select the Link Tool ()and click on the zone icon. The mouse pointer will change to a crosshair, indicating that a link is being created. Now click on the crack icon, selecting one of the red dots which appear as you approach the icon with the mouse (the red dots indicates where the links can connect to a given icon). The airflow in a link may go either way. The direction indicated by the link only defines the direction in which the flow is counted positively; flow in the opposite direction will be indicated by negative numbers in the output file.

A segmented black line connecting the zone and the crack, as shown below, should now be seen.

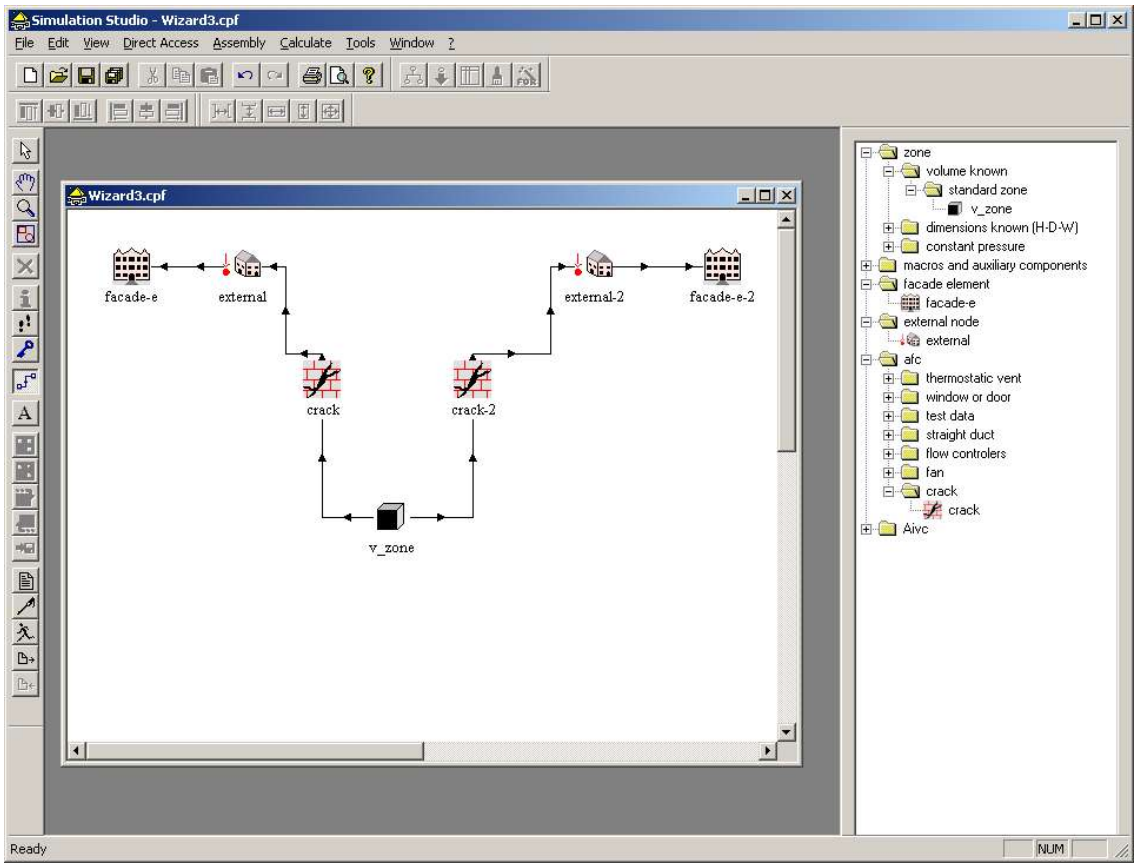



In the same way the following connections should be made:

- crack to external

- external to façade-e
- crack-2 to external-2
- external-2 to façade-e-2
- v_zone to crack-2

External nodes always come with a facade element. Therefore, a specialized macro model, containing an external node with its facade element, is also provided and could have been used instead. Macro models are discussed in more detail in the “Assembly Panel” section of the Simulation Studio User Manual.




*Assembly Panel Window with Components Linked Together*

## Running More Examples

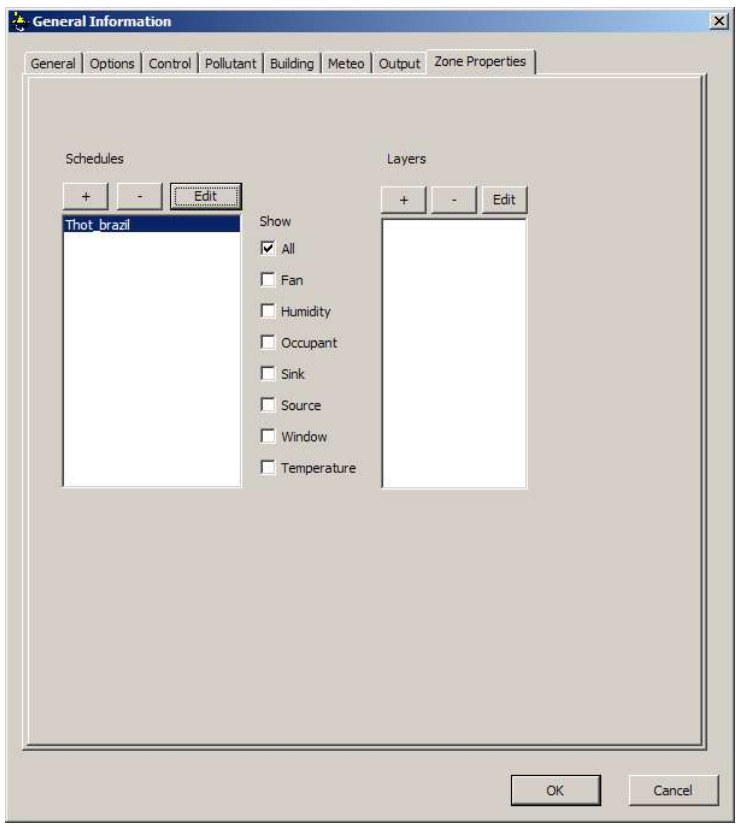
Several sample projects are provided in the standard distribution. To access these projects, use File/Open, and browse to the Examples directory. Change the file type to 'COMIS Project Files (*.cpf)' to view examples, select one and click the open button.

Several examples can be open at the same time. The Window menu allows to switch between several open projects, to view them at the same time, etc.

## Using Schedules

Some components, such as zones, allow for the definition of schedules to define events occurring during the simulation. Such events may concern : meteorological changes, occupant behavior (moving from room to room, opening windows, ...), pollutant emission/absorption, etc.

Schedules can be defined in the Global Control Window ( ), under the 'Properties' tab.



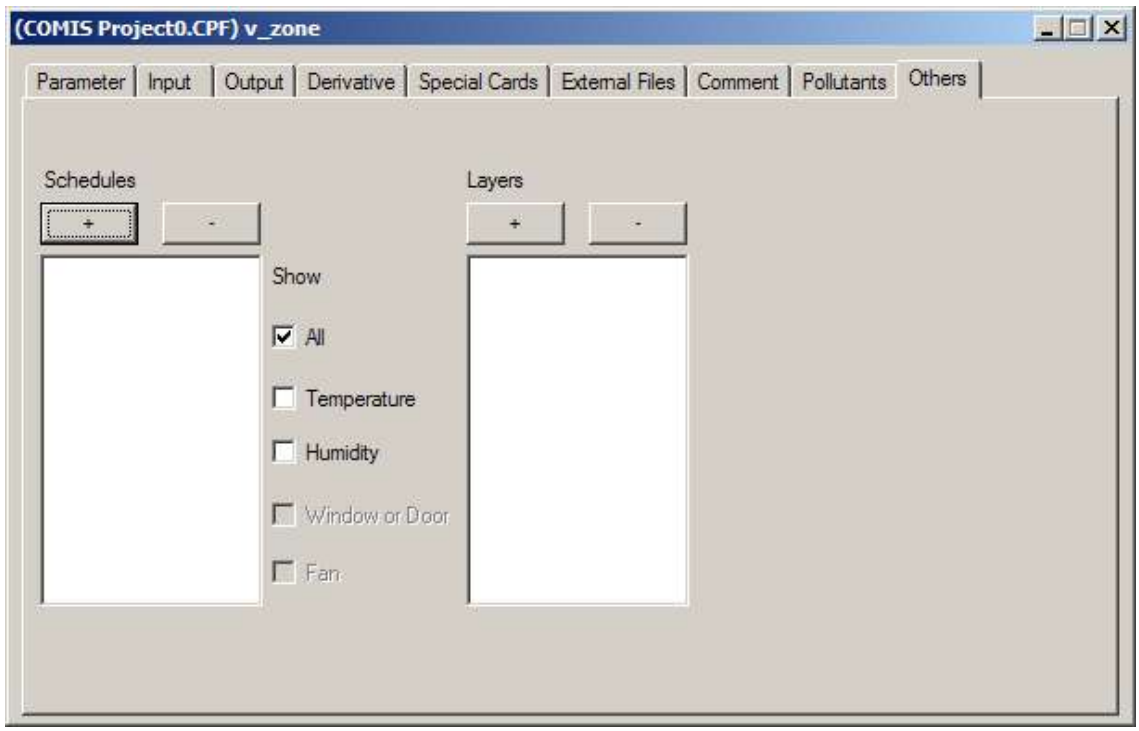
Creating a temperature schedule

In this window, you can use the '+' buttons to create, the '-' buttons to delete, and the edit button to edit schedules. It also allows to create zone layers. When creating a

schedule with the '+' button, the user has to specify an ASCII text file to store the schedule values.

Once created, these schedules (and layers) can be used with a given compoennt, such as a zone, by double-clicking on that component in the assembly panel, and selecting the 'Others' tab. Again, there are '+' and '-' buttons to add existing schedules to the component.

To find out what schedules can be attached to a given component, click on the 'Others' tab of the component's window. For example, when double-clicking on the zone icon in your sample project, a window similar to the one below will open.




*Attachments to zones*

# Other Simulation Studio Tasks

This concludes the introduction to the COMIS Simulation Studio. Users wishing to continue their COMIS education past the simple example given here should attempt the following tasks:

1. Look at the sample projects in the *Examples* subdirectory. Manipulate more complex components like windows, doors, fans, etc. The 'restaurant' example is prepared for coupling with the thermal simulation tool TRNSYS, but can also be run in stand-alone mode.

2. Create a more realistic example, using cp-values for the 4 main wind directions, such as the ones given in section 4.4 of the COMIS user Guide (under the description of the &-CP-VALUES keyword) :

| **Wind angle** | **0** | **90** | **180** | **270** |
|---|---|---|---|---|
| **Facade** | | | | |
| **4** | 1 | -0.49 | -0.45 | -0.49 |
| **3** | -0.49 | 1 | -0.49 | -0.45 |
| **2** | -0.45 | -0.49 | 1 | -0.49 |
| **1** | -0.49 | -0.45 | -0.49 | 1 |

   Set all zone temperatures to 20 degrees, as well as the outside temperature. Modify the wind direction during the simulation by defining a schedule in the lower part of the Meteo tab in the Control Card window.
   Observe the changes of air flows and be sure to understand their directions (From->To and To->From columns in the output file).

3. Study the COMIS User Guide and the Simulation Studio manual.

