CONTAM Demo

AE 552 – Air Quality in Buildings

PSU AE

October 23, 2012

Authored by: Jason W. DeGraw and Joseph Firrantello

CONTAM Demo

AE 552 – Air Quality in Buildings

Introduction

This demo is intended to give you an overview of CONTAM use. Given the flexibility of the program, it is not possible to give an in-depth tutorial on all aspects of the program – we simply do not have the time to do that. Instead, we'll go over some of the more important aspects and modes of use, and leave the more difficult parts for another time.

The Sketchpad Interface

The ContamW user interface is centered on the sketchpad, which is the large white area pictured in Figure 1. The sketchpad is a grid of icons – everything in the sketchpad is an icon, including walls. The sketchpad is not to scale, and only serves to orient the various icons that you will place in the sketchpad.

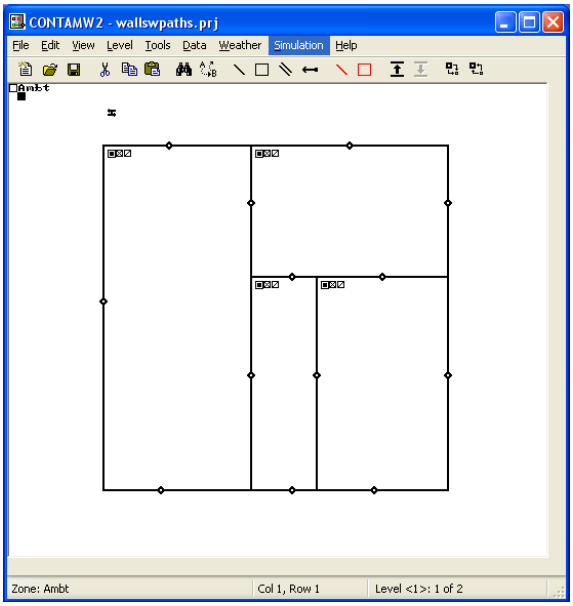

FIGURE 1 - THE CONTAMW SKETCHPAD

There are two modes of interaction with the sketchpad: a selection mode in which the cursor is a filled black box, and a drawing mode in which the cursor is an unfilled magenta square. In selection mode, the cursor is placed in a location by left clicking. Double clinking on some icons (e.g. zone and flow path icons) will bring up an icon specific dialog, and right clicking brings up a menu of possible actions. Walls are drawn in a Paint-like fashion using either the line tool ( ) or the box tool ( ). Walls will not be accepted unless a closed circuit is created, so the easiest path is usually the box tool (with selective deletions). Walls may be deleted by placing the selection cursor on the wall and pressing the delete key. To return to selection mode from drawing mode, right click on the sketchpad, press the escape key, or click the button of the drawing tool that you are currently using.

Caveat Utilitor

A number of warnings are in order. Some are obvious, others are not.

- **Save early and often** – ContamW operates upon a project file (with a prj extension) that is used as input for ContamX. This file is saved before a simulation is run and at various

other times without the user interaction. The lack of "undo" functionality means that if you make a mistake, this mistake may become a part of you project file and be difficult to undo. It is thus advisable to save at crucial points along the way as you develop your model (e.g. after you have finished wall input).

- **Be aware of units** – Many of the parameters that you specify have units, and the overall unit selection facility sometimes becomes confused (and/or confusing). Thus, always make sure that you have the correct units selected **before** putting in a number.
- **There is no undo** – In case you missed it above, there is no undo. In most cases, mistakes can be undone manually, but extra care beforehand is almost always going to end up being faster and less error prone.
- **Leave extra space** – Don't draw walls that are too short or zones that are too small. In order for icons to fit in a zone or on a wall, there must be room for them. A wall that is 3 units long will only have room for 3 icons, so be careful. Remember that the sketchpad is not to scale.
- **Don't forget the roof!** – Anecdotal evidence suggests that this is one of the common mistakes in CONTAM model development.

A Simple Example

The rest of this document gives step-by-step instructions on the construction of a simple model. In order to speed up the process, you should obtain the file startingpoint.prj, which contains a number of elements (airflow paths, sources, schedules, etc.).

Step 1: Start CONTAMW and load the project file

Locate CONTAM in the Programs menu (it should be under NIST) and start the program. You should see a window similar to the window in Figure 1 (without the walls and icons). Using **File→Open Project...**, navigate to wherever you have saved startingpoint.prj and open it.

Step 2: Set default flow units

For this problem, we'll use SI units of length and SCFM for flow units. Use **View→Options...** to bring up the Project Configuration Problems dialog. Change the default units of flow to SCFM, and then click **OK**.

Step 3: Draw the zones

Using the box and line drawing tools, draw the zonal arrangement shown in Figure 2. Remember that the sketchpad is not to scale, so the exact sizes are not important – only the relative locations of zones is important. Be careful to make all walls long enough that several icons will fit.

Step 4: Place zone icons

Right click in a zone, and select **Zone** from the pop-up menu. This places a reddish square icon in the zone. Double click on the new icon, and a properties dialog should appear. At a minimum, you will need to enter a zone name and a zonal area. The volume is computed using the level

height (which defaults to 3 meters). Set up four zones using the information in Figure 2. Save your work to a descriptively named file using **File→Save Project As...** (e.g. ae552-zones.prj).

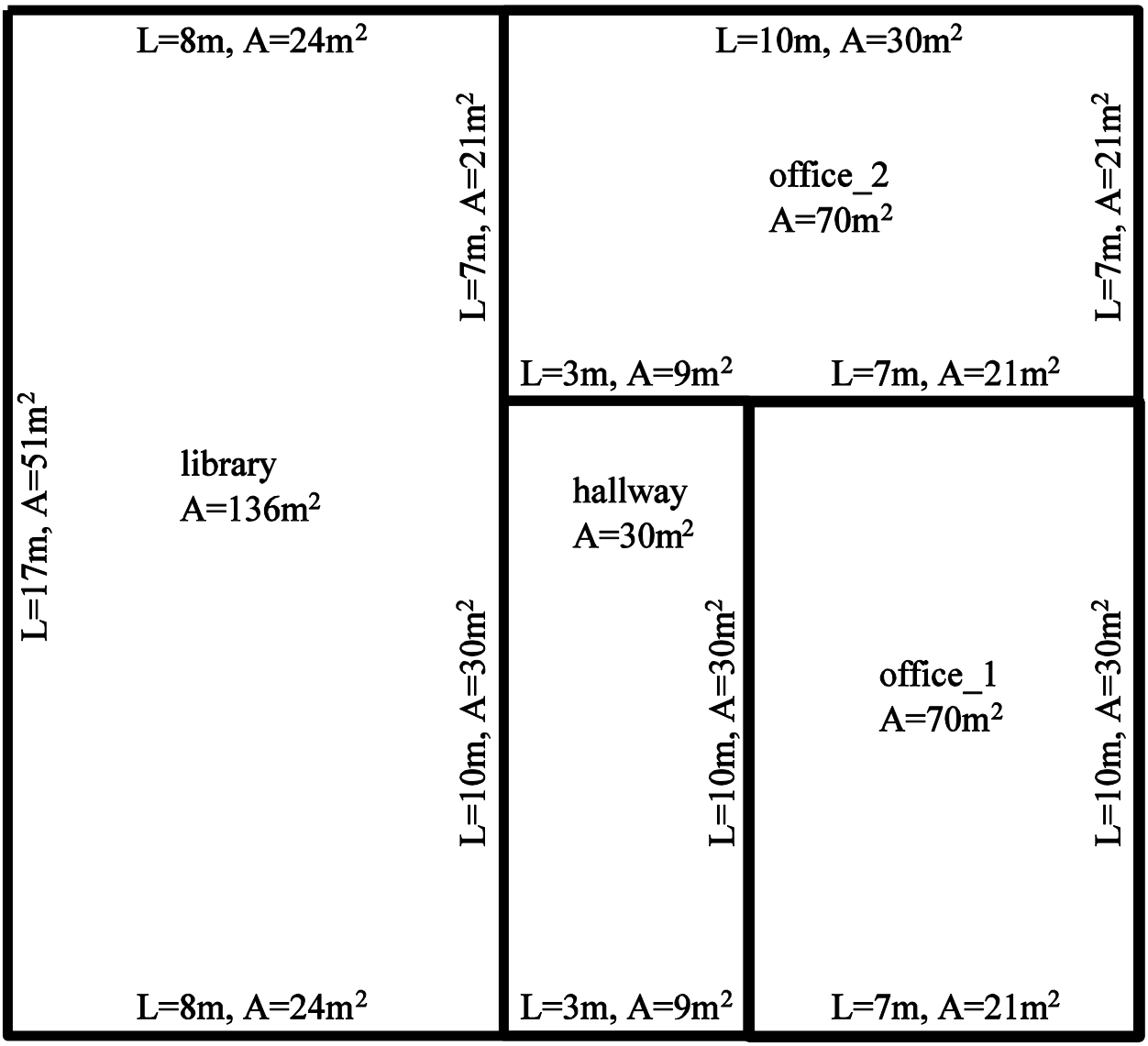

FIGURE 2 - A SIMPLE MODEL BUILDING

Step 5: Add exterior wall leakage paths

The easiest way to get the exterior wall flow paths is to set one up properly and then do a copy-and-paste operation to get the same airflow path onto each wall.

a) Right click on one of the exterior walls (which one is not important), and select **Flow Path** from the menu that pops up. A reddish diamond icon should appear on the wall.
b) Double click on this icon, and the Airflow Path Properties dialog should pop up.
c) From the **Name** drop-down menu, select **ExtWallAvg**. Information about the airflow path will appear in the **Model Summary** – in this case details of how the leakage is set.

d) Select the **Wind Pressure** tab, and for the **Wind Pressure Option**, select **Variable**. You will not be able to leave this tab until you select (or make) a wind profile. In the **Variable Pressure Data** section of the dialog box, select **low_rise_walls** from the **Name** drop-down menu. Click the **OK** button.
e) Copy this flow path to all other exterior walls. Position the cursor on the airflow path icon, and then press Ctrl-c (or use the toolbar copy icon). Place the cursor on another wall and press Ctrl-v (or use the toolbar paste icon). Repeat for all external walls.
f) The flowpath that we are using is essentially leakage per square meter, so we need to include this information in each flowpath. This is done using the multiplier, which is on the **Flow Path** tab of the Airflow Path Properties dialog (double click on a flow path to bring up the dialog). For example, the leftmost wall has an area of 51 square meters, so the multiplier should be set to 51. Set the multipliers for the wall leakage flow paths using the information in Figure 2.

Step 6: Add interior wall leakage paths

This process is very similar to the exterior wall case, but since there is no wind pressure to deal with the cut-and-paste process is not really needed. For each interior wall (there are five):

a) Create a flow path (right click, select **Flow Path**).
b) Double click the icon to bring up the properties dialog.
c) Select **IntWallAvg** from the **Name** drop-down menu.
d) On the **Flow Path** tab, input the correct multiplier (the number of square meters), and then click **OK**.

Step 7: Add a second level

In order to place our roof leakage paths, we need to add a second level. To do this, from the main menu select **Level→Insert Blank Level→Above current level**. You should be presented with a blank sketchpad for your second level. To navigate between levels, two buttons are available on the toolbar (↧↧). Also, **Level→Reveal Level Below** may be used to display the level below in gray.

Step 8: Add roof leakage paths

The process for adding roof leakage paths requires some care. The primary issue is that it is sometimes difficult to properly place the paths. The "reveal level below" feature is quite useful, but can become confusing in larger models. For this simple model it will work quite well. If you have not already done so, useuse **Level→Reveal Level Below** to show the walls on the previous level in gray.

a) On the second level, place the cursor "inside" one of the zones on the first level. Right click on the cursor location, and select **Flow Path**.
b) Double click the icon to bring up the properties dialog.
c) Select **RoofAvg** from the **Name** drop-down menu.

d) On the **Wind Pressure** tab, and for the **Wind Pressure Option**, select **Variable**. In the **Variable Pressure Data** section, select **roof** from the **Name** drop-down menu. Click the OK button.
e) On the **Flow Path** tab, set the **Relative Elevation** to zero. The default placement is at half of the level height, while we want the path right at the roof level.
f) Copy the flow path and then paste a copy of the path above each of the zones on the first level.
g) Set the multiplier of each flow path of each path to reflect the area of each zone (double click to bring up the properties dialog). For example, the hallway zone has an area of 30 square meters, so the multiplier is 30.

It is advisable to save again (e.g. ae552-paths.prj).

Step 9: Set wind parameters

Bring up the Wind and Weather Parameters dialog with **Weather→Edit Weather Data...** On the **Weather** tab, set the wind speed to 5 m/s, and then select the **Wind** tab. Set the **Roof or Wall Height** to 3 meters. Click **OK**. If a dialog pops up about modifiers for flow paths, click **Yes**.

Step 10: Create a simple air handling system

Right click in the ambient zone (outside of all the zones you have created), and select **Air Handling System** from the pop-up menu. Double click the new icon, which brings up a properties dialog with several tabs. We will only need the **AHS** tab. Give you air handling system a name (e.g. ahs), and then pick **30_percent** from the **Name** drop-down menu in the **Outdoor Air Schedule** section. This sets how much outdoor air will be pulled in by the system. Set a system volume under the **Supply System** and **Return System** tabs. CONTAM treats the volume *inside* the ductwork as a well-mixed zone, as there is (realistically) some delay from when a contaminant is taken in through the outdoor air intake and when it is delivered to the zones it serves. There isn't a rule of thumb for this, one typically has to find the average duct dimensions and multiply it by the total duct run length in a system. For the purpose of this exercise, set each to 2 m^2. Click **OK**.

Step 11: Add a supply and return to each zone

We want to supply each zone with 1 SCFM for each square foot of floor space, and 95% of this is to be returned to the air handling system. For each zone:

a) Compute the required supply and return airflows (in SCFM) using Figure 2.
b) Right click in the zone and select **Supply** from the pop-up menu.
c) Double click the icon and input the computed flow rate (check units!), then select the name of you air handling system from the **Name** drop-down menu.
d) Right click in the zone and select **Return** from the pop-up menu.
e) Double click the icon and input the computed flow rate (check units!), then select the name of you air handling system from the **Name** drop-down menu.

Save your work (e.g. ae552-ahs.prj)

Intermission: Solve the airflow problem

Before continuing on to the contaminant transport problem, let's solve the airflow problem. Use **Simulation→Set Simulation Parameters...** to bring up the simulation parameters dialog. We won't need to make any changes yet, but take a look at the various tabs and see what parameters are available. Click the **Run** button to close this dialog and bring up the ContamX control dialog. Click **Start Simulation** to run the simulation, and then **Close**. The sketchpad is now in results mode, and will display the results of your airflow simulation.

Before we can continue, we need to get the sketchpad back into normal mode – use **View→Normal Mode**.

Step 13: Turn on CO_2 as a contaminant

As it currently stands, no contaminants are set up for calculation. To change this, bring up the Project Species dialog with **Data→Contaminants...** Select carbon dioxide in the Species box, and then click **Edit**. Check the **Use in Simulation** checkbox, and then click **OK** twice.

Step 14: Add CO_2 sources

To approximate the presence of occupants, we will add sources that inject the correct quantity of CO_2 into the zones.

a) Right click in zone office_1, and select **Source/Sink** from the pop-up menu.
b) Double click the icon to bring up the properties dialog.
c) Select **CO2_one_person** from the **Name** drop-down menu.
d) On the Multiplier, Schedule and Location tab, set the multiplier to 2. Click **OK**.

This source will stand in for two occupants. Repeat the process in zone office_2 (multiplier 1) and in zone library (multiplier 3).

Step 15: Set simulation parameters

Open the simulation parameters dialog with **Simulation→Set Simulation Parameters...** Change the simulation method for contaminants to **Transient**, and then change the simulation time step output to 5 minutes. Click **OK**, and then save your work (e.g. ae552-done.prj).

Step 16: Run simulation

Run your simulation with **Simulation→Run Simulation**, then click **Start Simulation** to run the simulation, and then **Close**. The sketchpad is now in results mode, and will display the results of your airflow simulation.

Step 17: Postprocess results

In order to look at our results, there are a couple of options. Within ContamW, we can plot results using **Simulation→Plot Contaminant Result...**, which will generate on-screen plots of the contaminant. The dialog is fairly self explananatory, so spend some time trying different

things. You can also export your results to a text file that can be read into a spreadsheet with **Simulation→Export Results...** This process is also fairly self-explanatory, but be careful when you are reading the text file into a spreadsheet or you will end up with one column. Additionally, stand-alone programs CONTAMRV and SIMREAD can be used to postprocess your results. Using ContamW, see if you can answer the following questions:

a) Which zone ends up with the highest carbon dioxide concentration?
b) Which zone ends up with the lowest carbon dioxide concentration?
c) How long does it take for the building to reach steady-state conditions?