

IDA ICE 5.0 – Climate (with stratification) model Tutorial

This document presents a short tutorial on how to use the new **Climate (with stratification) model** in IDA ICE 5.0. It is assumed that the user is familiar with previous versions of IDA ICE.

Instructions

Step 1: Create the model

We begin the tutorial by creating a building with a room. To this aim we press the *Start new building with a single zone* button.

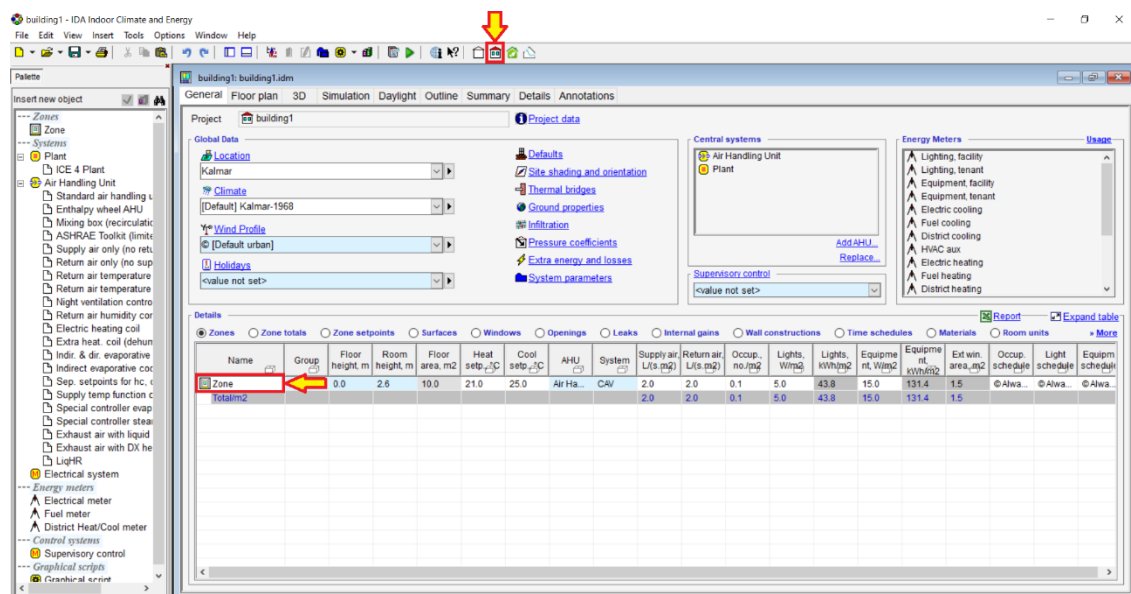


Figure 1: The IDA interface with the "Start new building with a single zone" button, and the General tab with the Zone details.

Then, in the *General* tab we double click on *Zone* in order to open the zone's window. Here, we select the *Advanced* tab and we make the following choices:

- We set the *Model fidelity* field to *Climate (with stratification)*.
- In the *Air stratification* field, we choose the *Layer height* to set the size of the air layers (i.e., the vertical discretization). Notice that decreasing excessively the layer height inevitably leads to slower computations! (i.e., choose wisely the vertical discretization).
- In the *Grid for results* field, we select the space and time resolution for the 3D visualization of the results on a uniform measurement grid. Alternatively, we can choose to visualize the results only in a "Measuring volume" within the zone.

- In the *Measures* field, we choose which variables to visualize. The available variables are: Air temperature, Air velocity, Mean radiant temperature, Operative temperature, Predicted Mean Vote, Predicted Percentage of Dissatisfied, Predicted Percentage of Dissatisfied due to Draught, CO₂ fraction and Illuminance. Note that fine grid resolutions together with high sampling frequencies lead to computationally slow simulations and large result files!
- If *Measuring planes* and *volumes* are added to the floor, these are shown in the respective field. They are *Active* by default but can be easily deactivated by the user.

The fields described above are highlighted in Figure 2.

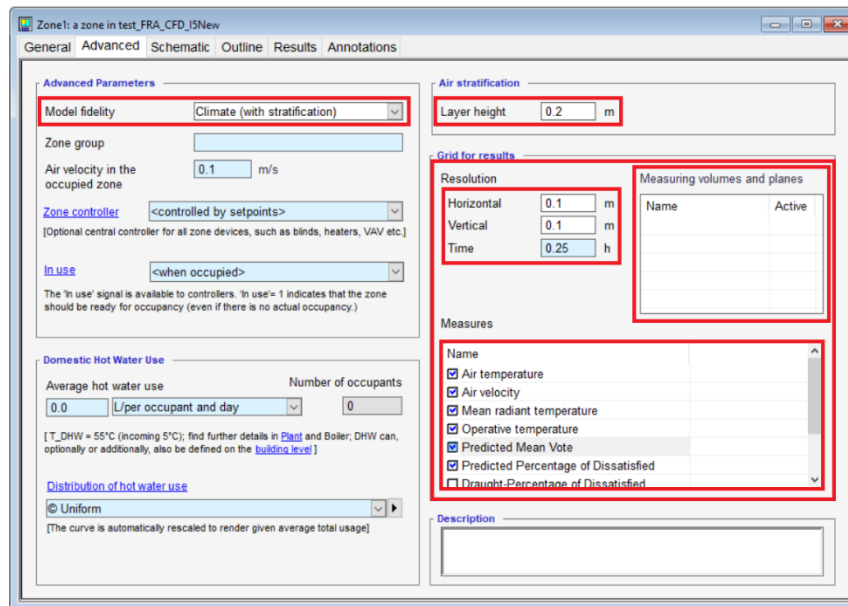


Figure 2: Advanced tab in IDA ICE 5. The fields highlighted in red are described in the main text.

The default zone has furniture, light, equipment, an occupant, an ideal cooler and heater, and an Air Handling Unit of type CAV. Notice that in the Climate (with stratification) model, the equipment and the occupant generate plumes, while a jet originates at the location of the Supply terminal of the Air Handling Unit. The Supply and Exhaust terminals, the equipment and the occupant all have default positions. To move the equipment, double click on it, go to the Outline tab and change its position (see Figure 3). The same can be done for the occupant. Alternatively, the user can open the floor tab and either move the occupant with the mouse or change its position in the Properties tab (on the left; see Figure 3). By default, all equipment and occupants that are added to the zone, will be treated as plumes. Finally, to locate supply and exhaust terminals, the user must open a surface and drag the terminal from the Palette on the left. By opening the Supply terminal, the user can choose the *Air Source* (i.e., to which AHU is the terminal connected), the terminal's *area*, *distance from surface*, *direction*, and *multiplicity*. The factor k is indicative of the jet expansion. It is assumed to be approximately 10 for pure round jets. For Exhaust terminals, the user can choose the *Air Source*, the *area*, and the *distance from surface*.

Other “Features” that give rise to “Flow Elements” are electric and water radiators, and heating/cooling panels. By default, these features generate modified wall-currents if located on the lateral walls, or plumes if located on the floor or ceiling. The user can also drag the “Electric radiator” feature in the “Room Units” field. By double clicking in the radiator, the user can also import an

external geometry that represents the actual shape of the radiator. In this case, the radiator will be treated as a plume.

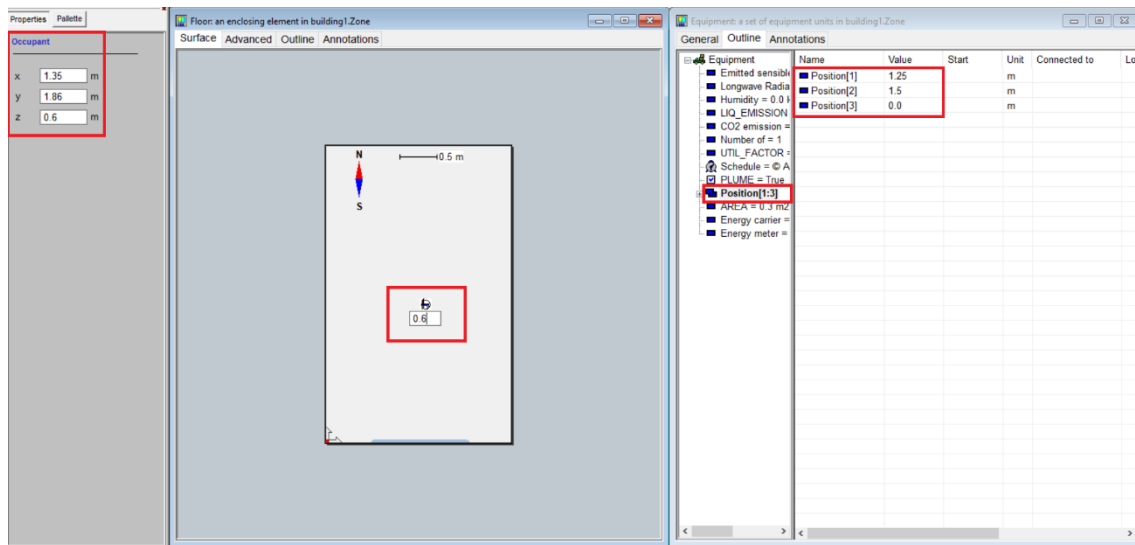


Figure 3: The floor with an occupant (whose location is reported in the Properties tab on the left), and the Outline of the equipment (where its position is highlighted).

The user can now perform directly a simulation by choosing one of the options in the *Simulation* tab (see Figure 4). Alternatively, the user can “Build” the model and work in the “Advanced Level”.

Step 2: Simulate and visualize the results

For standard simulations, simply press the simulation button. When working in the “Advanced Level”, perform the simulation by clicking on Tools -> Run model, or by pressing the Run button in the Advanced Level field of the Simulation tab.

Wait for the simulation to complete.

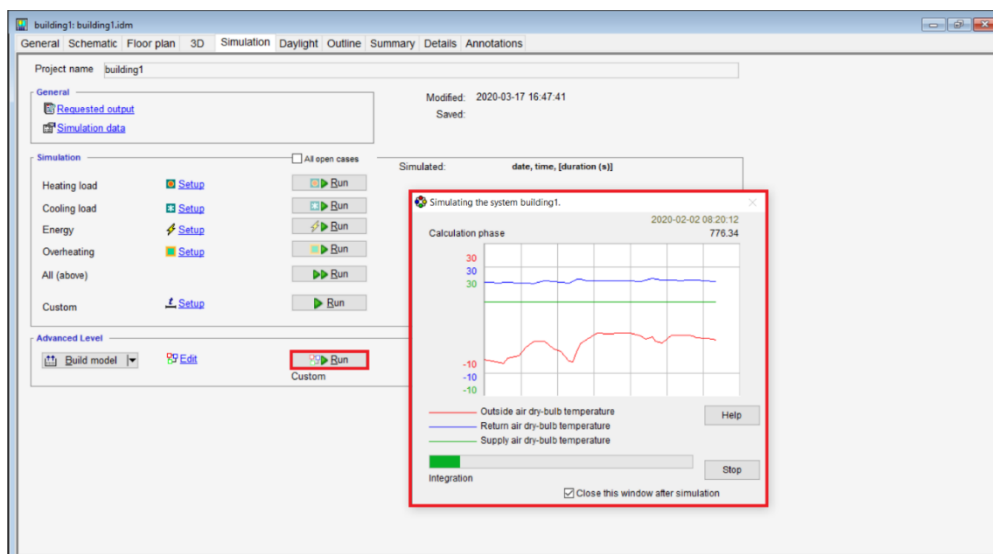


Figure 4: Ongoing simulation with the Climate (with stratification) model.

Notice that it is now possible to see the evolution of the variables during the simulation by opening the components in the Schematic view (Advanced Level only). This was not possible in IDA ICE 4.8.

In order to visualize the 3D variables' fields, the user needs to:

- Select the 3D tab in the model window and press the *Animation* button.
- Choose one of the measured fields among those selected in step 1.
- Click Show (see Figure 5).

Optionally, explicit min and max values may be selected in the *Range of values* field. The calculated measurement field is displayed in the zone. The x+,x-,y+,y-,z+ or z- buttons may be used to view the results of a plane of choice. CTRL + Left mouse button can be used to move the plane. In the same way, the small red line in the scale can be dragged in order to change the iso-line/iso-surface.

To conclude, we shall write a few considerations on the 3D results.

The most valuable information that the user can obtain from this new zonal model, is the result in each horizontal layer. The 3D view of results is obtained by post-processing the data. This means that grid points within a layer, will get its temperature and absolute velocity (by default equal to zero if there is not a flow element). Then, at the locations of room features, windows and flow elements (including wall-currents), the results will be accordingly modified in the corresponding grid points. For example, within a jet the temperature and velocity of the grid points will be modified. This is done by assuming that the temperature and velocity fields perpendicular to the jet trajectory, have Gaussian profiles. Hence, temperature and velocities are reconstructed from the trajectory values (at the centerline). This means that 3D results are qualitative rather than quantitative. However, the user can still get a very good idea of the thermal comfort in each room location.

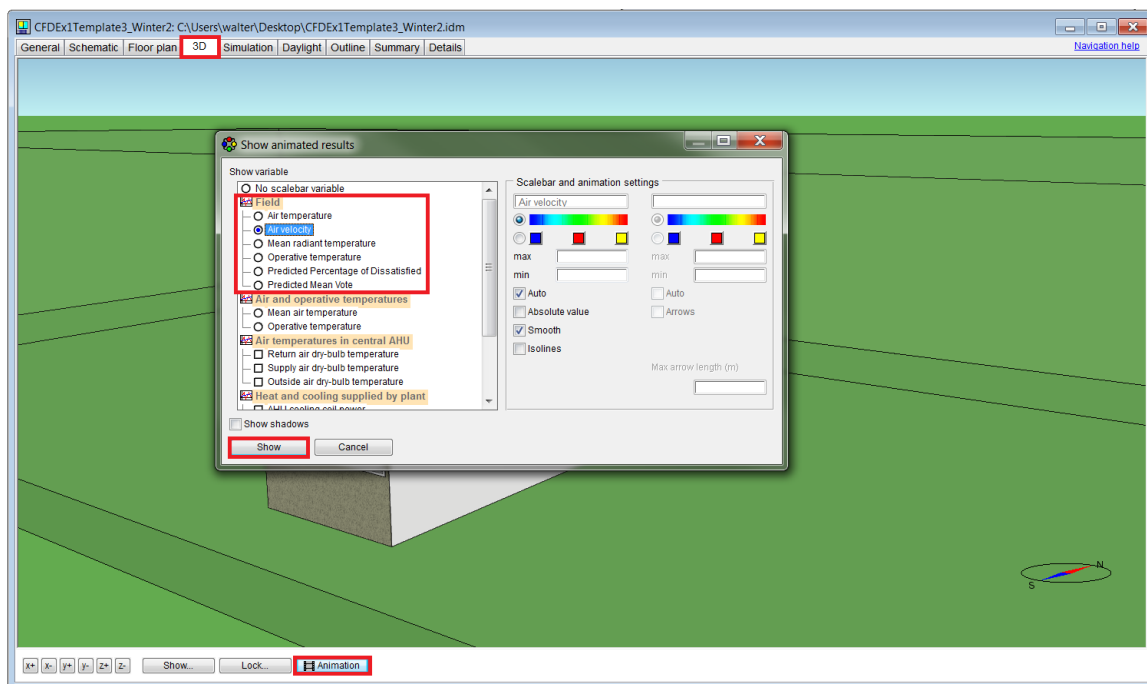


Figure 5: Steps to visualize the 3D fields.

Optional: Working in the Advanced Level

To work in the Advanced Level, the user needs to “Build” the model by selecting Tools -> Build Model (see Figure 6). Once the model is built, the *Schematic* tab appears. By clicking on it, we have access to all the components in our system, from the boilers and chillers in the standard plant, to the detailed window in the zone (see Figure 7).

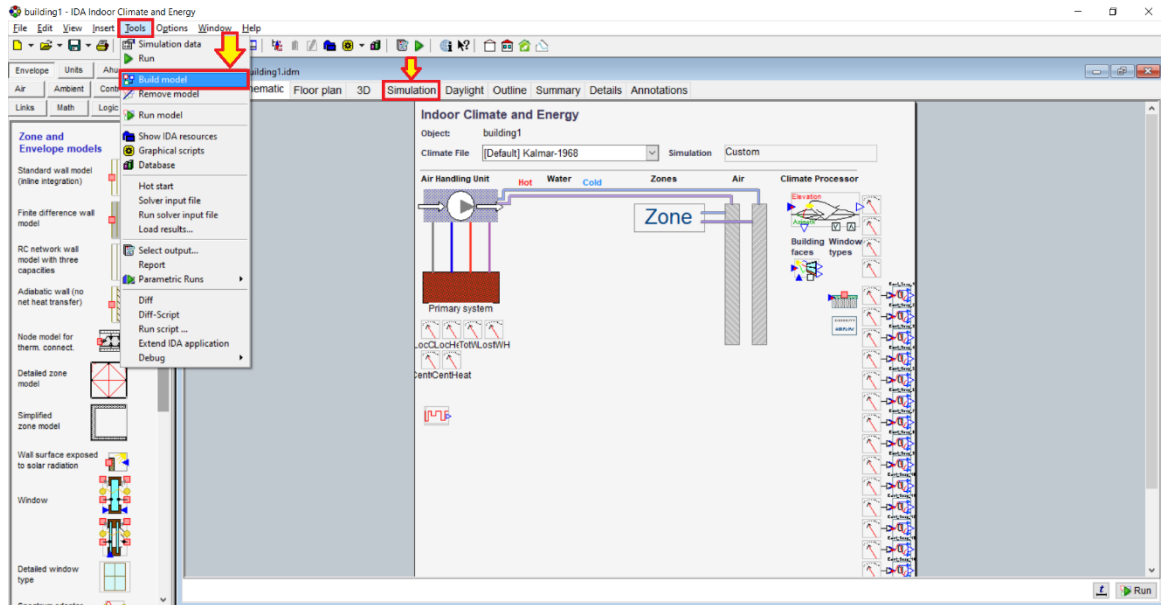


Figure 6: The schematic view of the model created by clicking on Tools -> Build model.

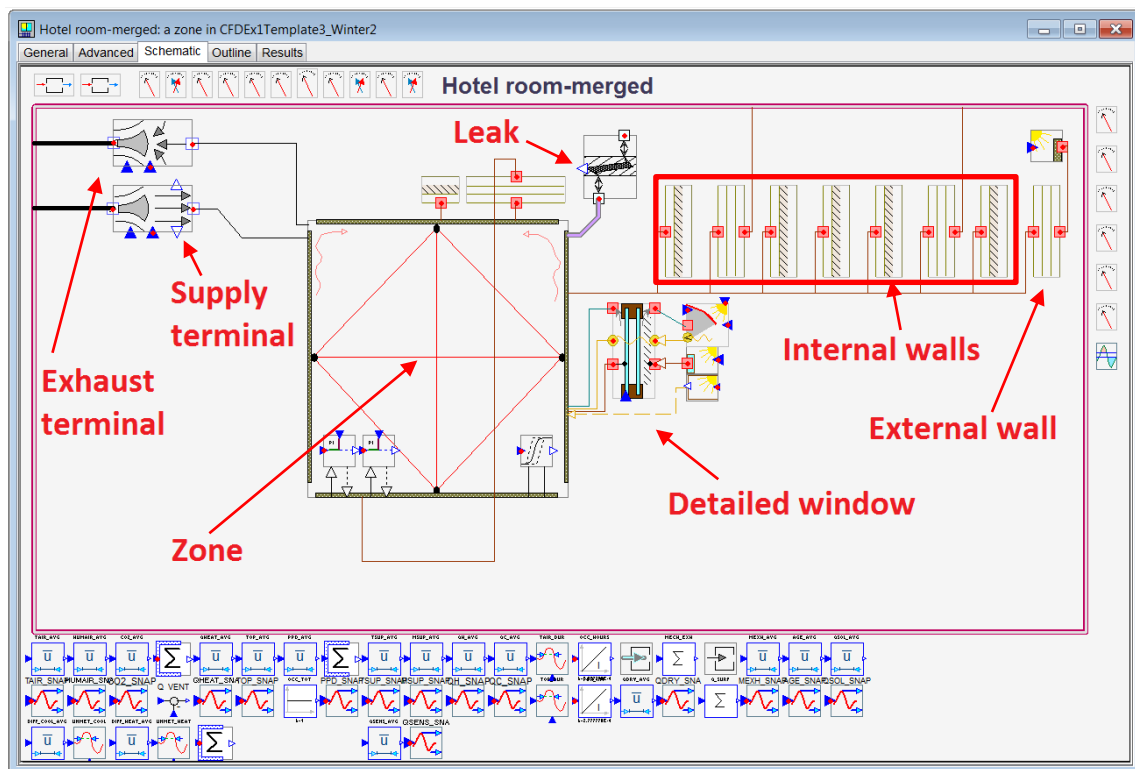


Figure 7: Schematic view of the Detailed zone.

