Vayu Pravah User Manual

वायु प्रवाह

November 2021

Version 1.0.27

BEEP
BUILDING ENERGY EFFICIENCY PROJECT

Disclaimer:

November, 2021

Contents

1 Introduction

In India less than 10% of the urban residential dwellings are actively cooled. Affordable housing, which is following a rapid development path, should be planned with the best possible natural ventilation potential. Generally, this aspect is not successfully addressed as no low cost and easy to use tools of this kind are available. With the objective to actively support proper designs, a freely distributable tool for analyzing wind driven air movement between buildings for natural ventilation has been developed by BEEP. It is a “real comprehensive” Computational Fluid Dynamics software (based on OpenFOAM, the best public domain software in this domain) which can be distributed freely. Equivalent commercial software’s do generally cost in the range of ₹ 4 to 20 lacs for one license.

Vayu Pravah has been developed in collaboration with the University of Applied Science Western Switzerland (HES-SO Valais) a free CFD (Computational Fluid Dynamics) tool for assessing wind driven air movement between buildings for large residential projects and/or local municipal planning. It has been compared/validated against wind tunnel experiments and other expensive commercial software. Its user interface has been developed for non-specialist users like architects, HVAC engineers, urban planners.

The beta version of the tool has been tested successfully by the Indian BEEP team and by other stakeholders.
The potential use of this software goes beyond India and is relevant for all the dense urban tropical areas. SEforAll has already shown interest in promoting its use in other countries.

1.1 Background about the software development of Vayu Pravah (वायु प्रवाह)

Vayu Pravah (वायु प्रवाह) is a simulation environment tool using OpenFOAM with an environment specially developed for non CFD specialists.

OpenFOAM® is the leading free, [open source software](#) for [computational fluid dynamics (CFD)](#), owned by the [OpenFOAM Foundation](#) and distributed exclusively under the [General Public Licence (GPL)](#). The GPL gives users the freedom to modify and redistribute the software and a guarantee of continued free use[1].

The scientific methodology approach for the version 1.0 has been published (see below).

IOPscience Journal of Physics: Conference Series

Development of a freely distributable CFD tool for approximate and detailed simulations of the flow around a complex of buildingsDevelopment of a freely distributable CFD tool for approximate and detailed simulations of the flow around a complex of buildings

J. Decaix* (HES-SO), P. Jaboyedoff (Effinart), G. Duthé (ETHZ), E. El Sergany (EPFL Master Student), L. Aiulfi (ESRI) , CISBAT 2021 – Lausanne, Switzerland Carbon Neutral Cities - Energy Efficiency & Renewables in the Digital Era EPFL, 8-10 September 2021

https://iopscience.iop.org/article/10.1088/1742-6596/2042/1/012069

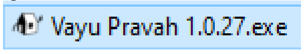
[1] https://cfd.direct/openfoam/about/

2 Installation

2.1 System requirements

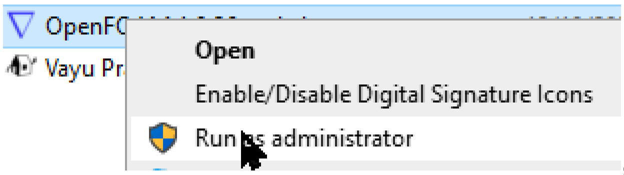
The software can be installed on Windows 10 systems (and has also been tested on Windows 8.1.)

The Vayu Pravah installation package consists of the **OpenFOAM 1.0.26 and above.exe** 

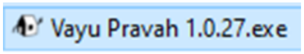
, and **Vayu Pravah exe** 

files.

a) To install the software, first run the OpenFOAM 1.0.26 and above needs to be installed as system administrator



,

b) followed by the Vayu Pravah exe file 

.

All the files (including project files) should be saved under the C drive in the system.

Any previous version of Vayu Pravah in the system must be deleted before any new installation.

Figure 2-1 Virtualization and Hyper V status

If it is not, virtualization must be allowed in the BIOS or UEFI. After making these changes, the machine should be restarted.

After these settings have been made, the software package can be installed.

3 Functional user guide

This part of the manual deals with creating a new file of a building design, adjusting various parameters to prepare a simulation, running a simulation, and visualizing the results.

3.1 Home Page and File Input

3.1.1 Open or Create a Project

The user can initiate a project by directly clicking on the Browse option and loading the file. The home screen of the software also has the list of recent projects which can be reopened by clicking on the project name. The user can also specify another project to load by clicking on 'Open project'.

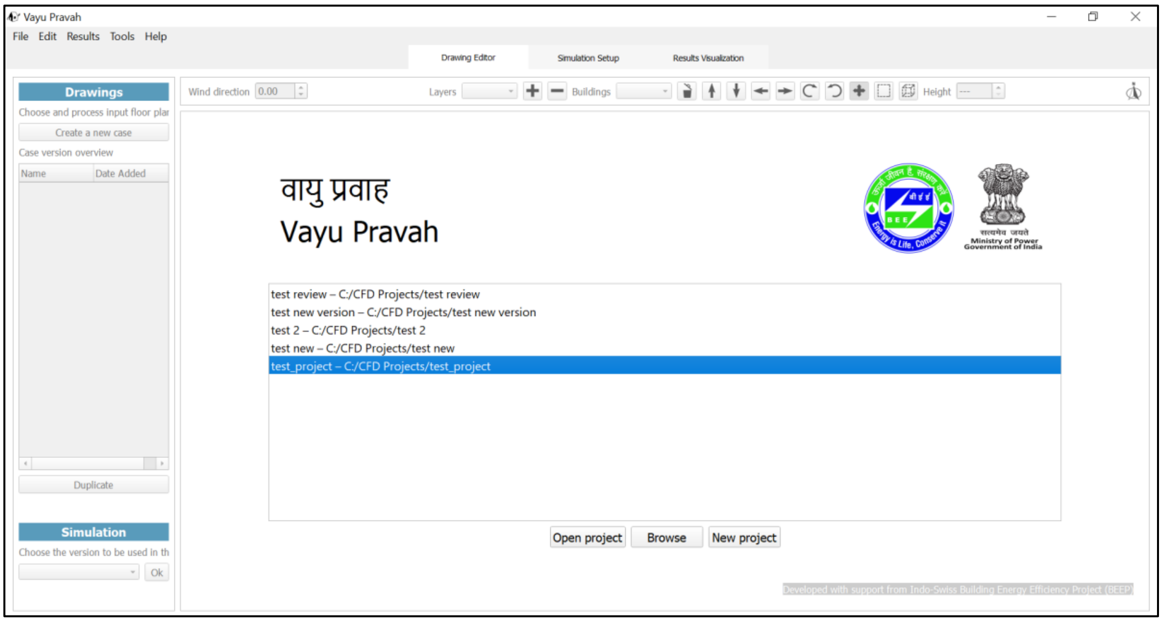


Figure 3-1 Home Page: create a new project or open an existing project

3.1.1.1 Creating and saving a new case

A new case is added by clicking on 'Create new case'. The user is first prompted to save the case with a name, providing a brief description, and then start working on it by adding different layers.

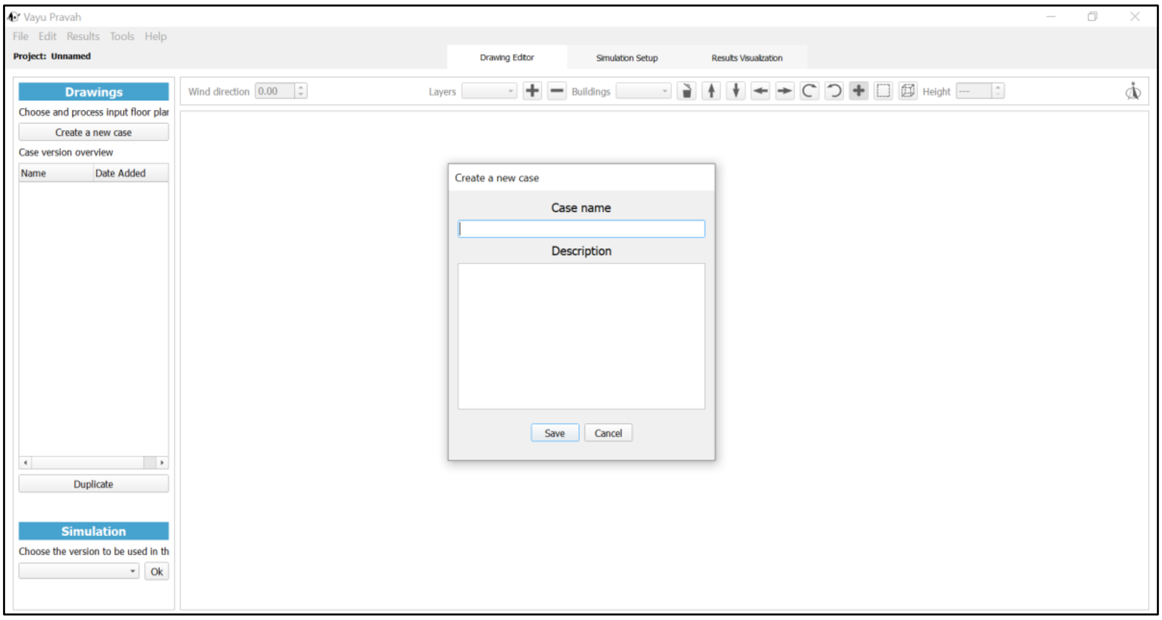


Figure 3-2 'Create a new case' prompts the user to name and save a new case

3.1.1.2 Importing a dxf (or a part thereof)

Once a case of the project has been created, the user can begin adding the dxf files by adding ‘Layers’ (explained in detail later). This is done by clicking on the ‘+’ icon next to the Layers in the Drawing Editor.

The dxf file should be a 2 D drawing with dimensions measured in meters (not feet/mm/inches). Also, the drawing should be cleaned before loading to remove internal elements like furniture etc from the plan.

The polygon representing a plan view of a building layer must be closed.

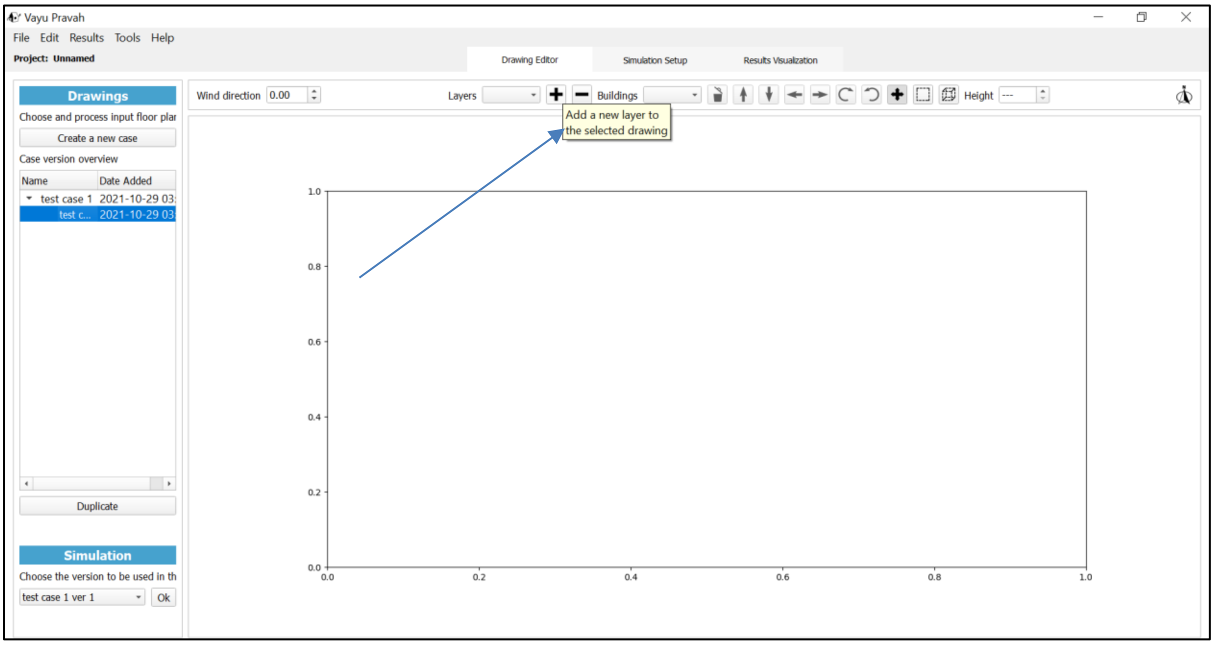


Figure 3-3 Importing a dxf file for one layer

A building which is monolithic will consist of one layer only. If the contour changes over the height, then layers must be added accordingly.

The user can click on a part of the drawing when importing to include or exclude it from the plan. By default, all the drawing in the dxf is selected, but the user can manually select or un-select part of the dxf to be input in the plan. The selected part of the drawing is highlighted in blue (against the grey input file).

3.1.1.3 Importing a dxf with selections of building dxf components

The user can choose to select a part of a dxf building or the entire building (as shown in example below).

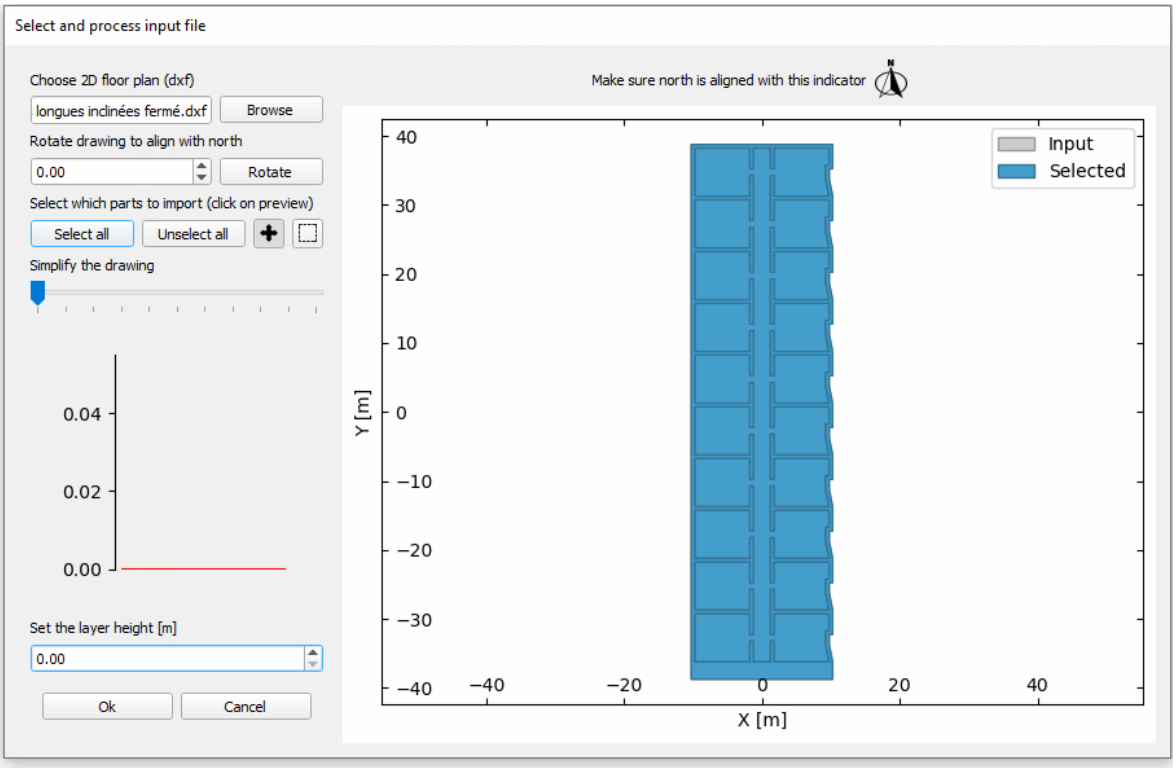


Figure 3-4 Importing the entire building (in blue)

Or the user can choose to select only part of the building by clicking on it.

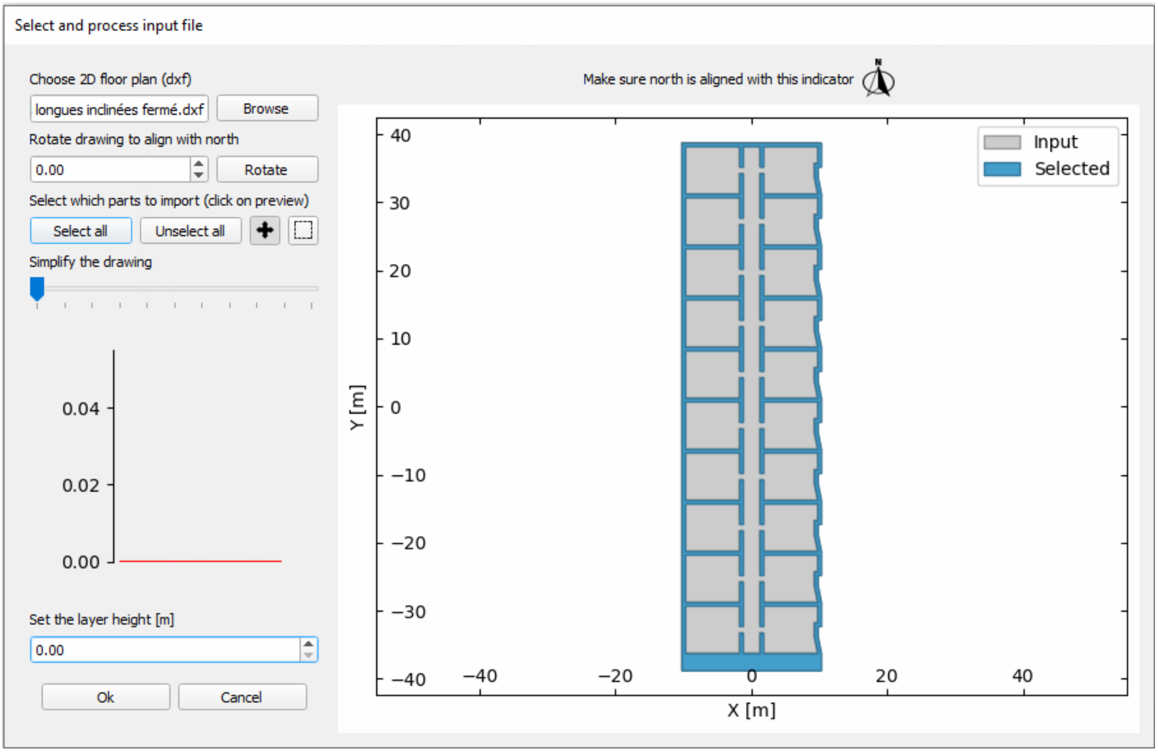


Figure 3-5 Selecting only a part of the dxf

3.1.1.4 Building selection

Similarly, in case of multiple buildings, the user can select any number of buildings for the plan.

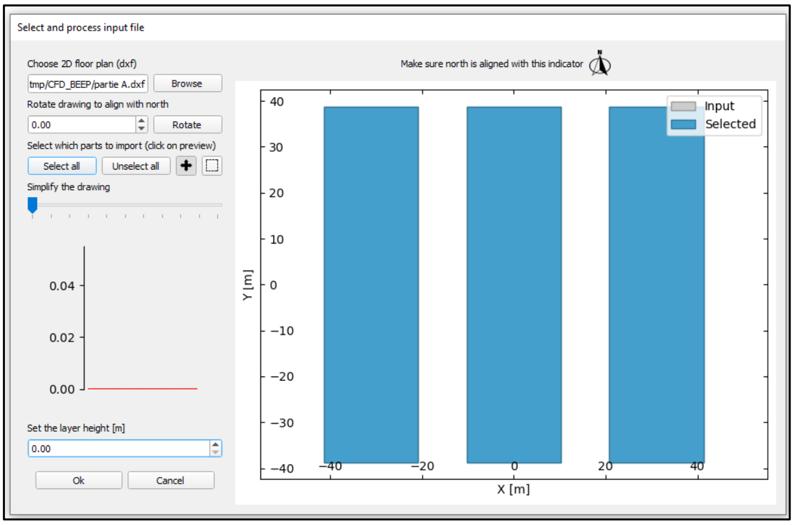


Figure 3-6 Loading a new file: By default, all the buildings have been selected (in blue)

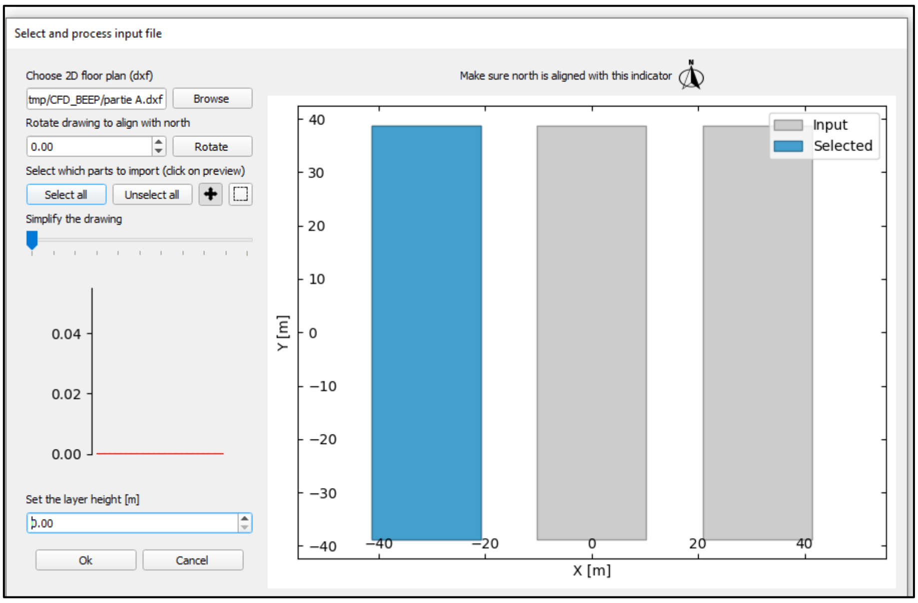


Figure 3-7 Manually selecting part of the dxf to import in the plan

The entire project consists of multiple dxf 'Layers' which can be at different heights from the ground. At the time of loading the file, the user can enter the height of the layer from the ground. This is represented by the red line in the graph.

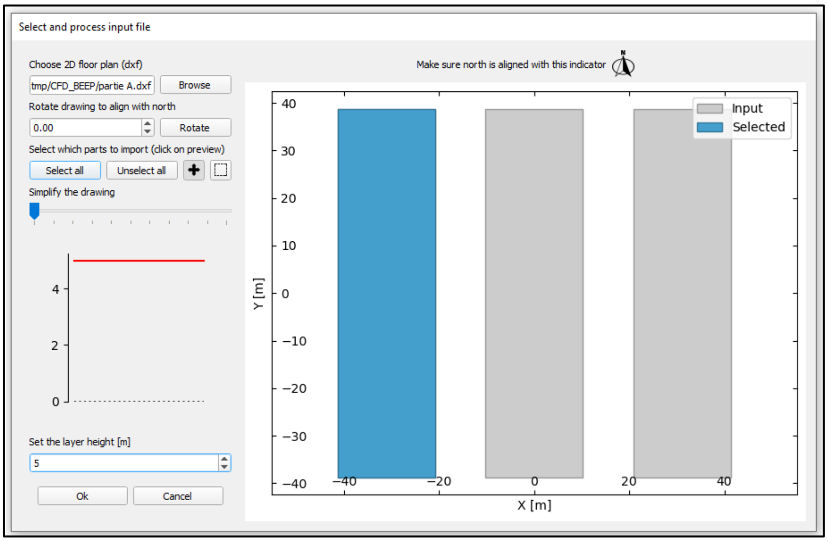


Figure 3-8 The red line represents the height of the layer from the ground

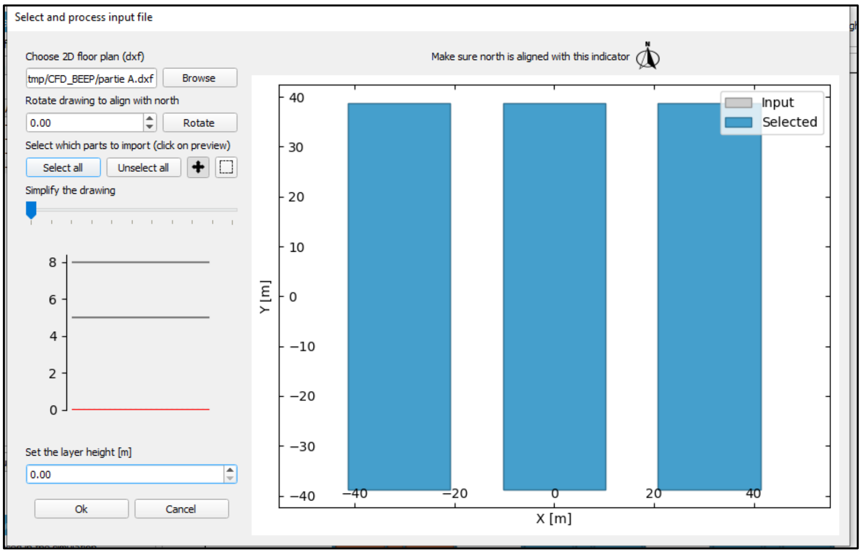


Figure 3-9 Entering layer height (shown by the red line in the graph); all existing layers in the project are shown by black lines on the same graph

3.2 Basic use of the Drawing Editor

Once the building(s) has been loaded in the drawing editor, the user can modify and edit it in the Drawing Editor.

3.2.1 Selecting a building

In a particular drawing, the user can select a building by clicking on it. The selected building in the dxf file is highlighted in brown as contrast to the unselected buildings in blue. It is on this building that all the changes are done.

By default, the building no. 1 is selected.

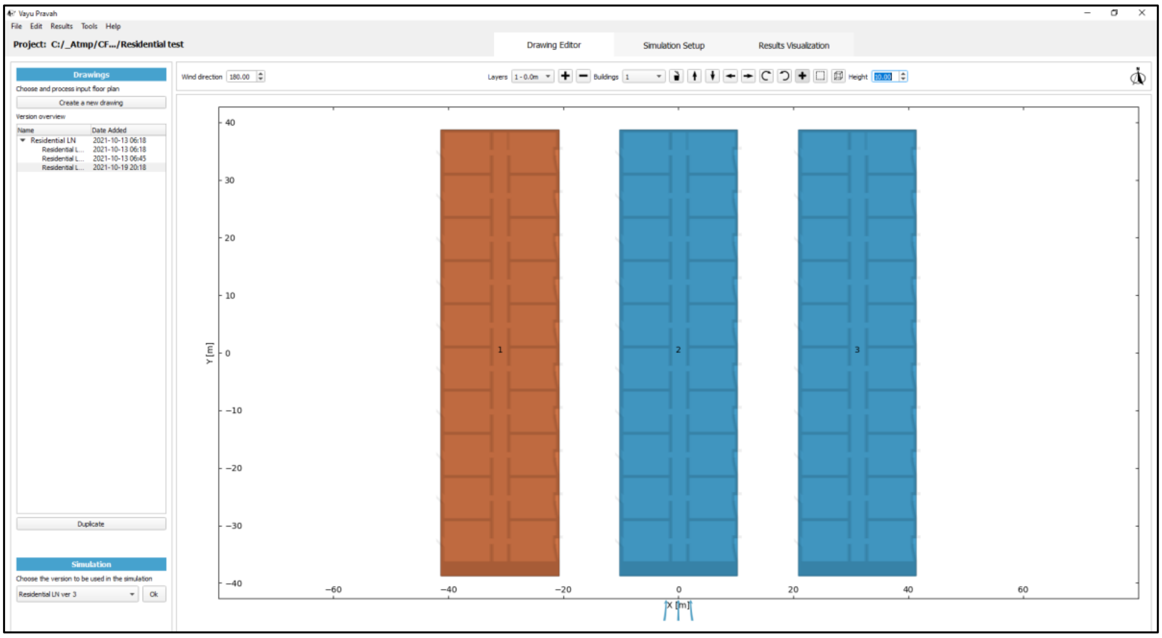
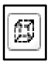


Figure 3-10 Building 1 is selected and highlighted

3.2.2 Selecting multiple buildings

The user can select multiple buildings in a layer at once to work on by clicking on the '' icon and moving over the desired area in the drawing editor. All the buildings in this area will be selected.

To select multiple buildings in different layers, click on the '' icon. This will select all the buildings in the highlighted area in all the layers.

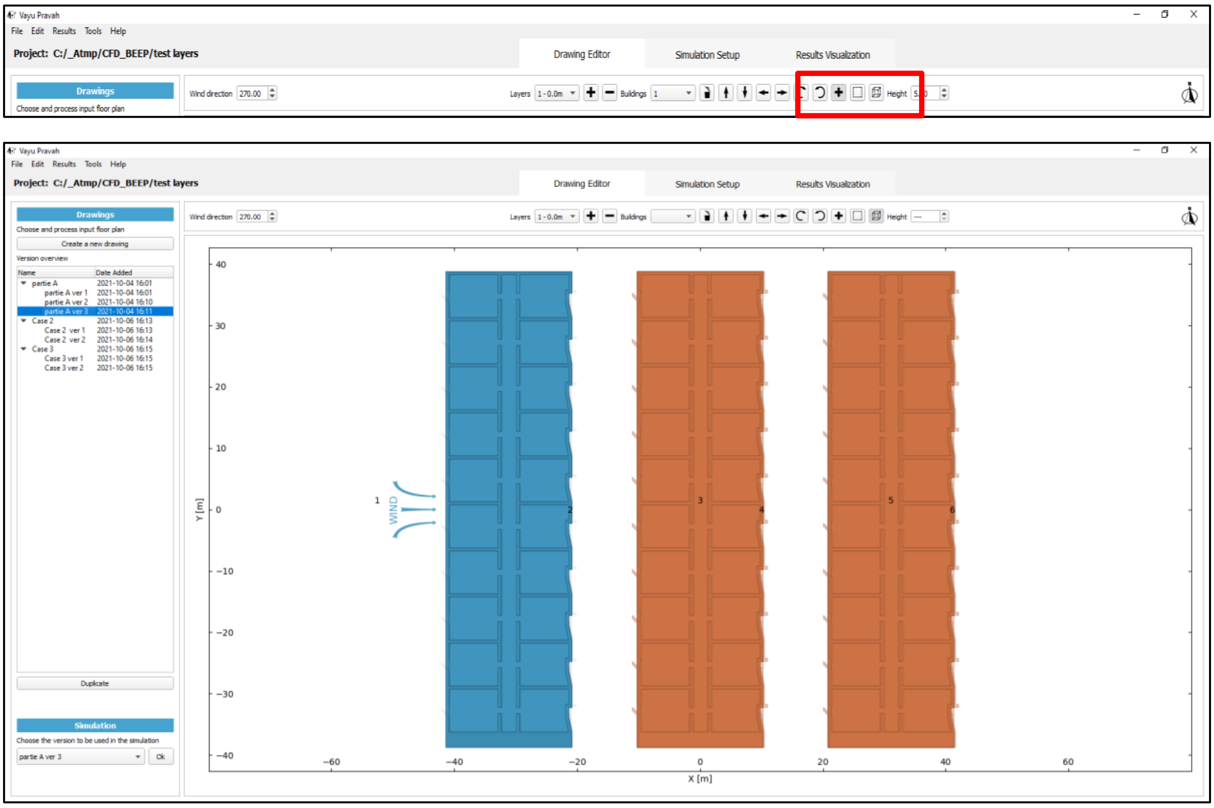


Figure 3-11 Selecting multiple buildings in one layer

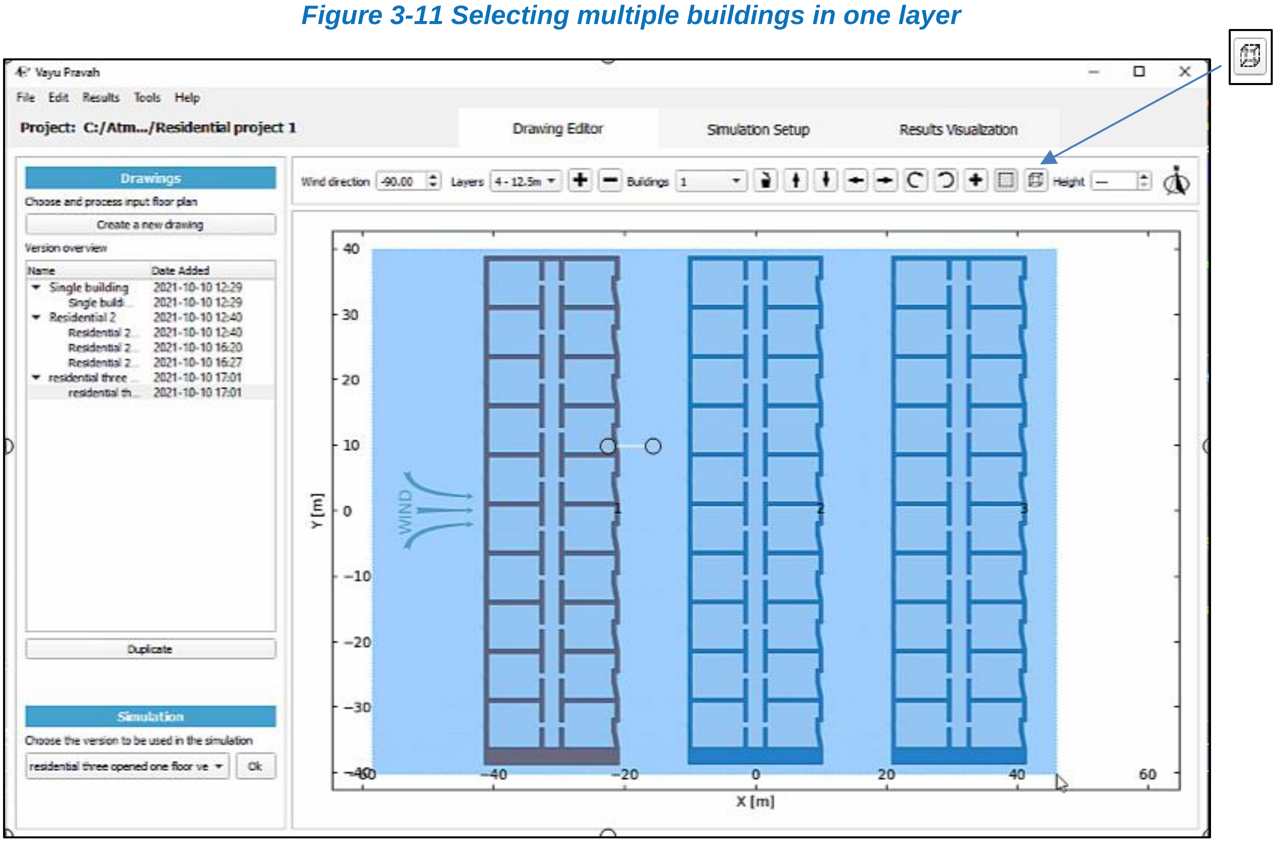


Figure 3-12 Selecting multiple buildings across all layers

3.2.3 Enter building heights

The user can enter the height of a building layer in the drawing editor by clicking on the building and entering its height in the 'Height' option on the top right. This ensures that the heights are correctly mapped and is particularly useful when the number of buildings is large.

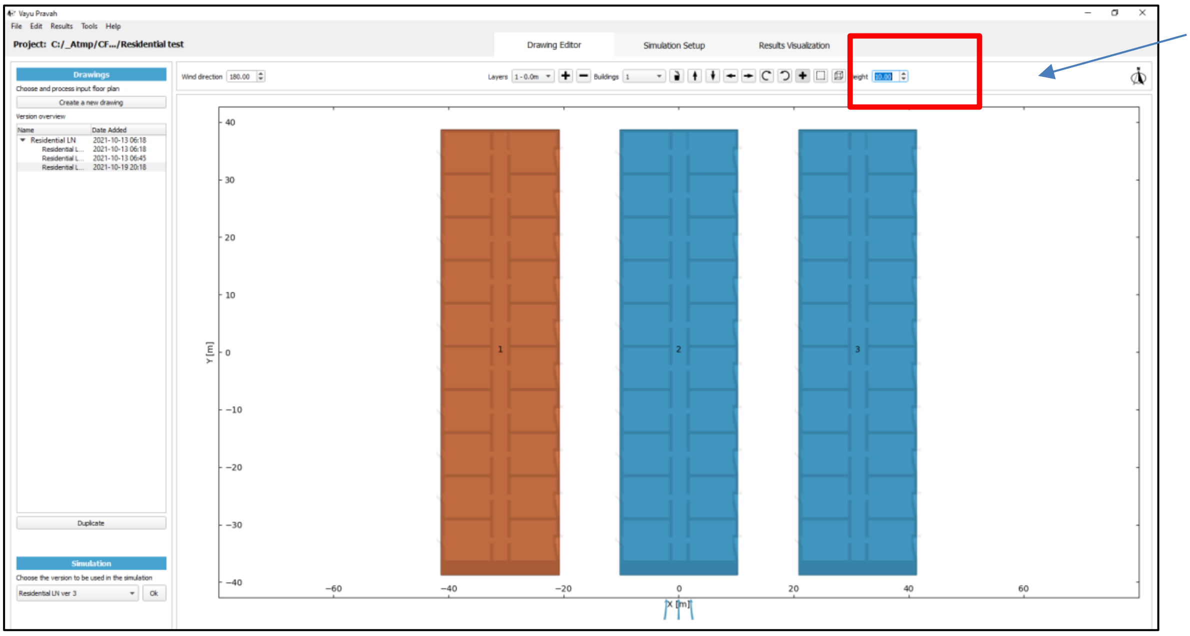


Figure 3-13 Entering building heights

3.2.4 Deleting a building

A building can be deleted by selecting it and clicking on the '' icon next to it.

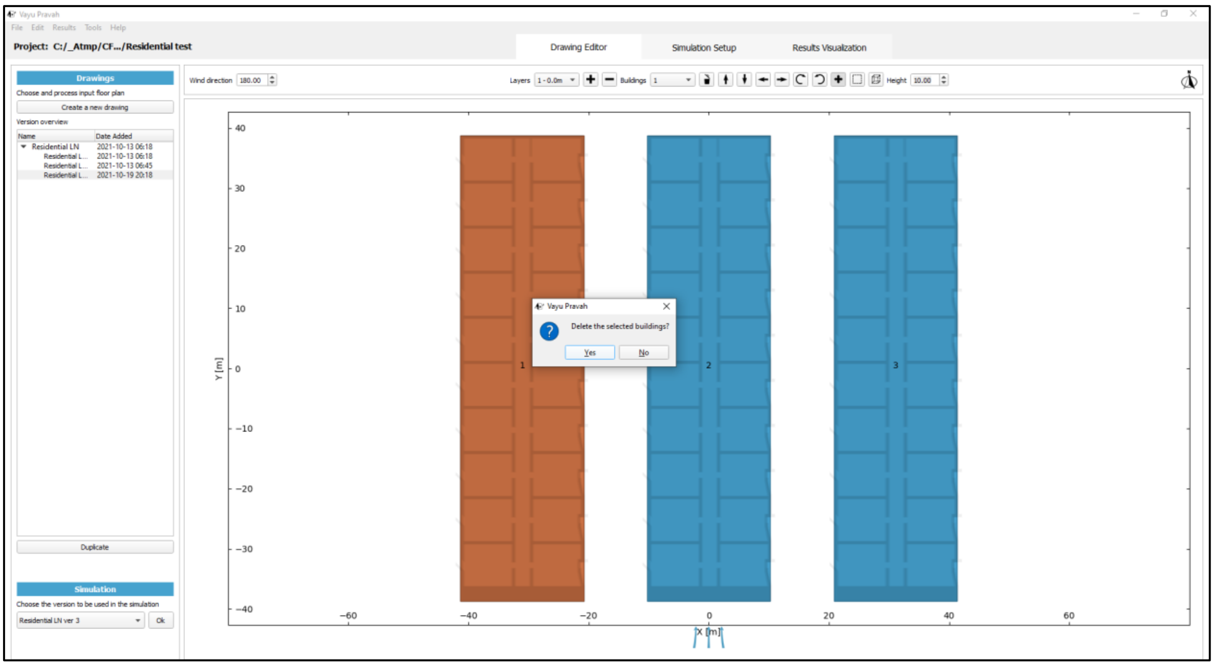


Figure 3-14 Selecting a building to delete

3.2.5 Adjusting view

The view of the plan can be moved using the ‘’ icon.

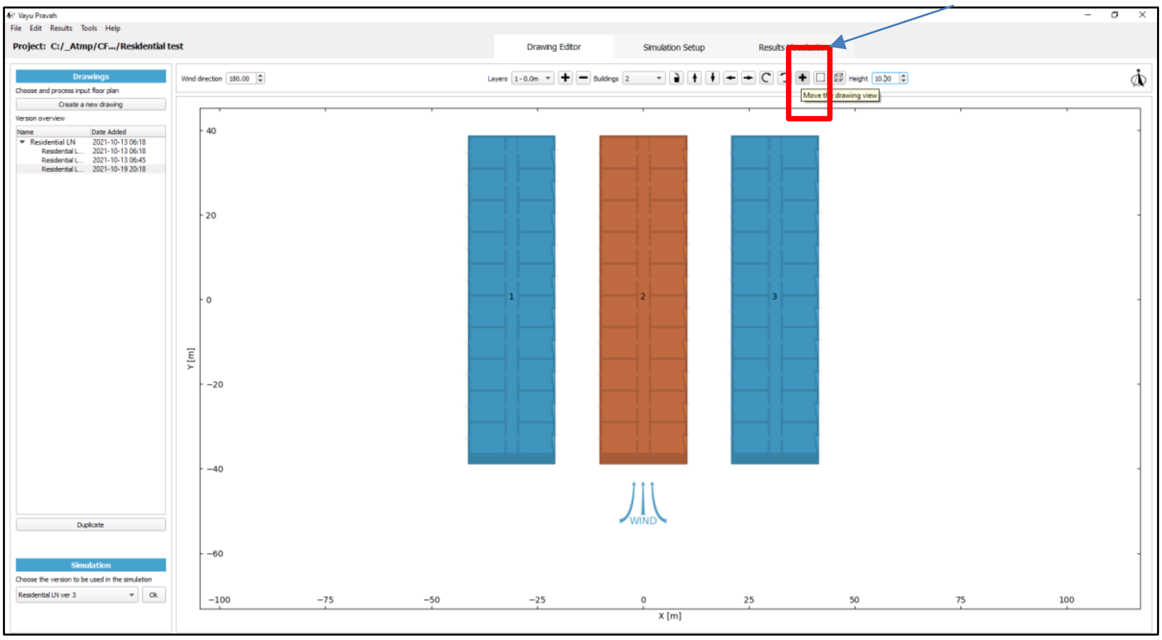


Figure 3-15 Moving the drawing view for selected building

3.2.6 Creation of ‘Layers’

A project in the Vayu Pravah consists of different layers of a building wherein the user can represent several floors of a building, different façade profiles (like windows etc) and visualize the internal flows. The layers are added as separate dxf drawings of individual layers and added at specified heights. An example of layers is presented below.

3.2.6.1 Original plans with the different facades and openings

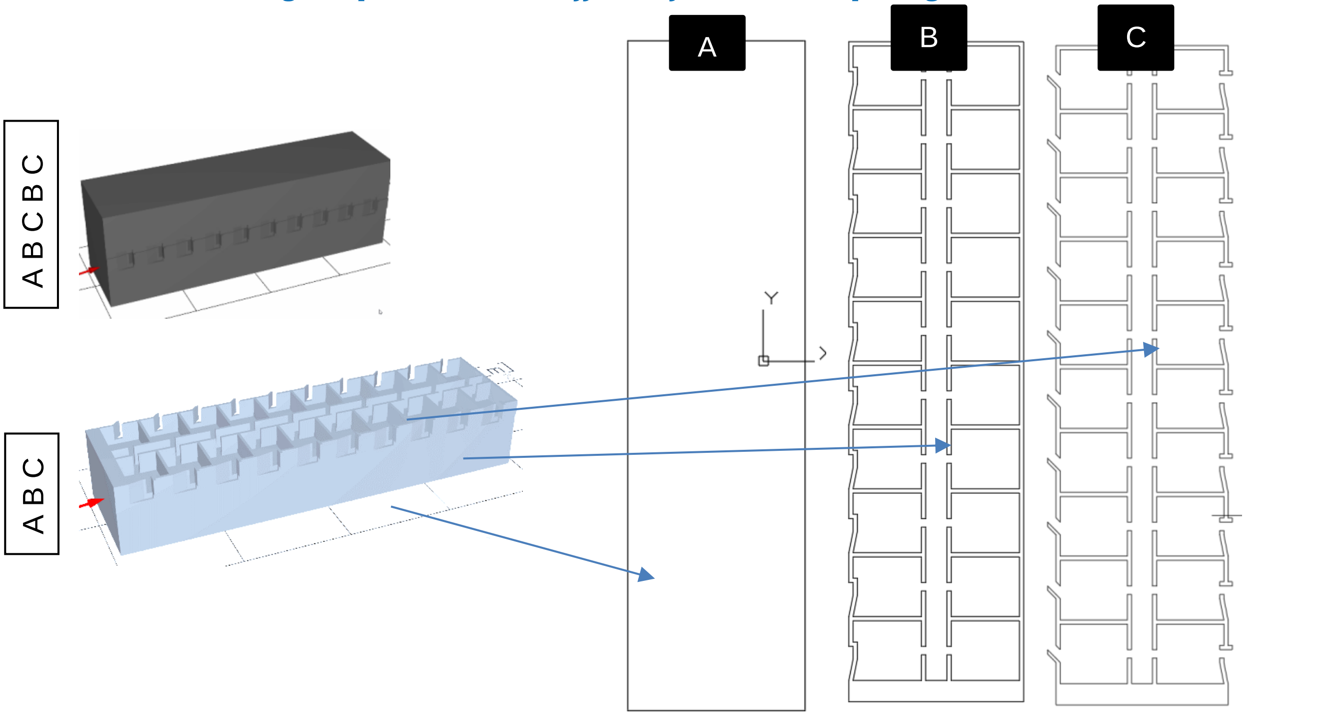
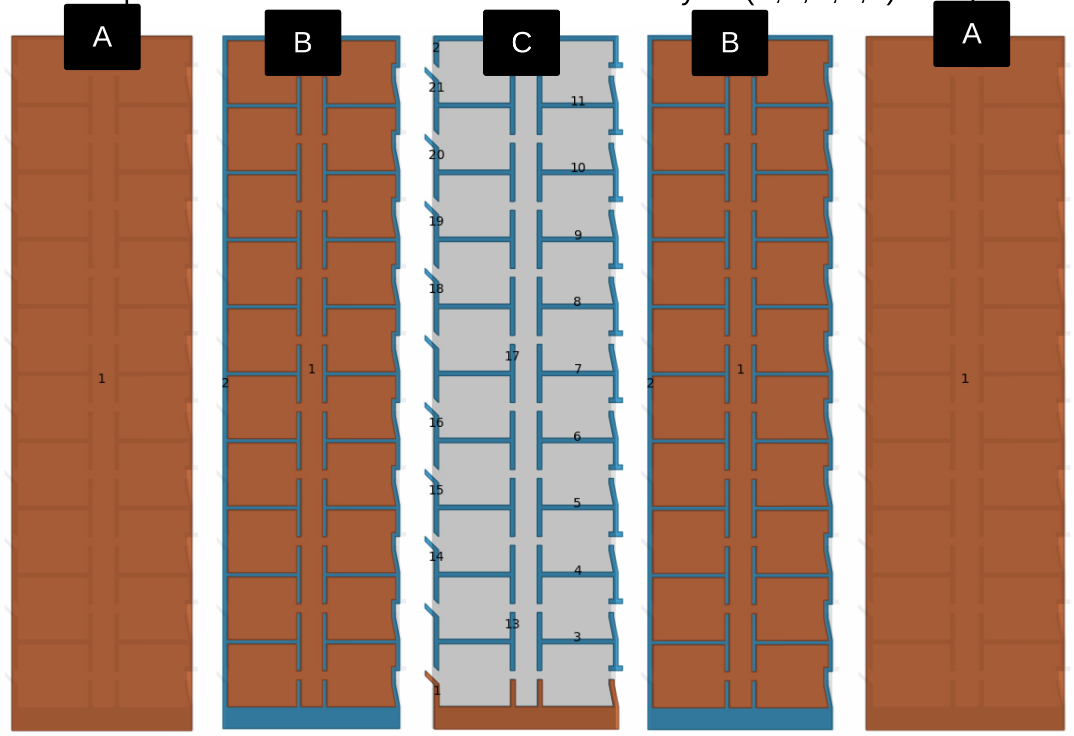


Figure 3-16 : façade profiles A) Bottom and top Rectangular closed façade profile B) closed façade with patterns C) open façade with aerodynamic elements

For the example above, we are using 3 different layers A (bottom and top), B (closed profiled façade), and C (profiled façade with opening and aerodynamic elements) to construct a building with straight and closed facades for the bottom and top part.

3.2.7 Adding a layer

The example above would be constructed with five layers (A,B,C,B,A) on top of each other.



A layer is added by clicking on the '+' on the top panel. This will prompt the user to add a new dxf drawing as the new layer. The input process remains the same as described earlier. The height of the layer can be specified here. It will be shown in the graph with a red line. Any preexisting layers are denoted by black lines on the same scale.

3.2.8 Moving between layers

The user can choose the specific layer to work on from the drop-down menu of Layers.

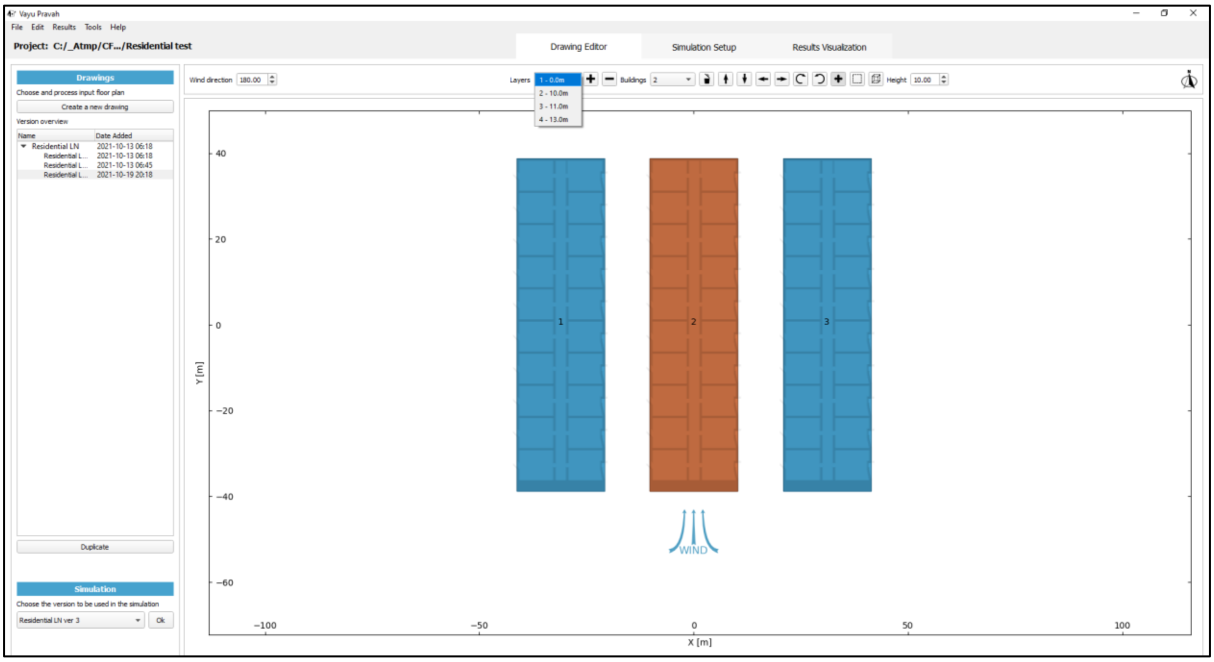


Figure 3-17 Switching between different layers by the drop-down menu

3.2.9 Deleting a layer

An existing layer can be deleted by selecting it from the Layers drop down menu and clicking on '-'.

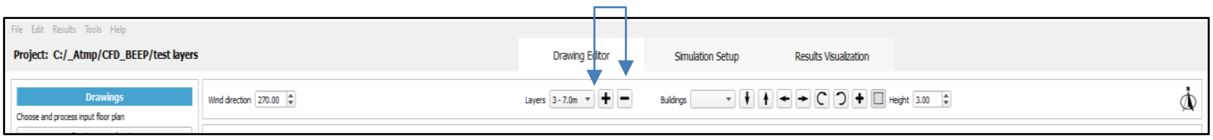


Figure 3-18 Adding and deleting layers

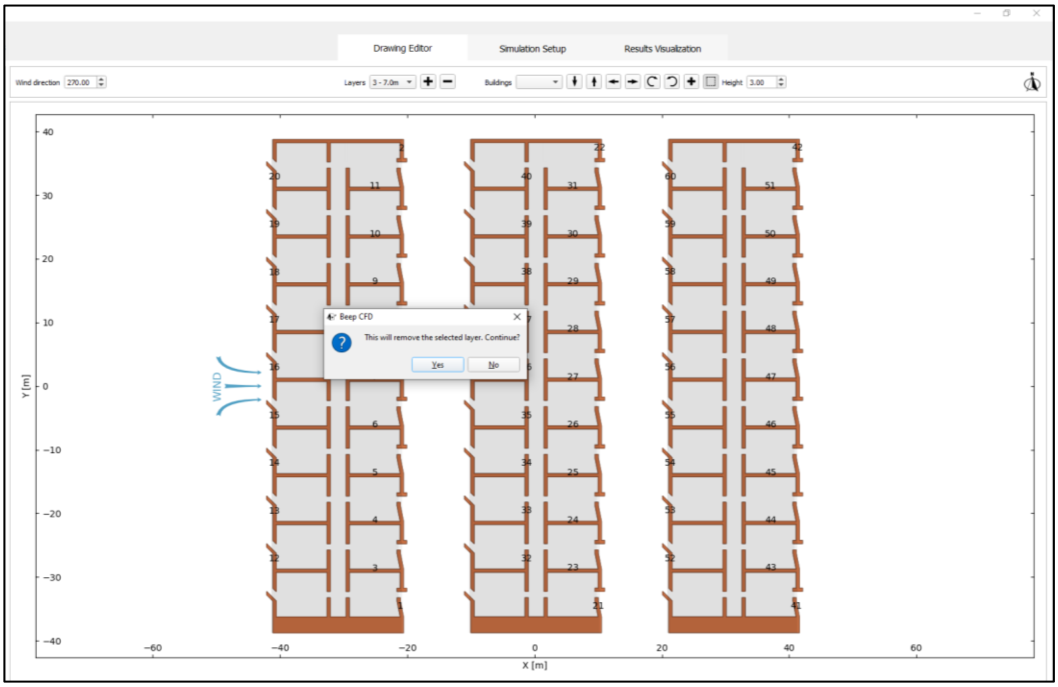


Figure 3-19 Deleting a layer by clicking on '-' icon next to Layers

3.2.10 Project, case, and version control

In the Vayu Pravah software, each project can consist of multiple cases (i.e. different design files), and all the variations done in a particular case are saved as different versions. The user can select any version of any case to work on in the current project by selecting it from the 'Case and version overview' panel on the left of the drawing editor.

3.2.10.1 Duplicate versions

In the version control, one has the option to duplicate a version to create a new copy and save it as another version.

Clicking on Duplicate copies the version and creates a duplicate as a new version.

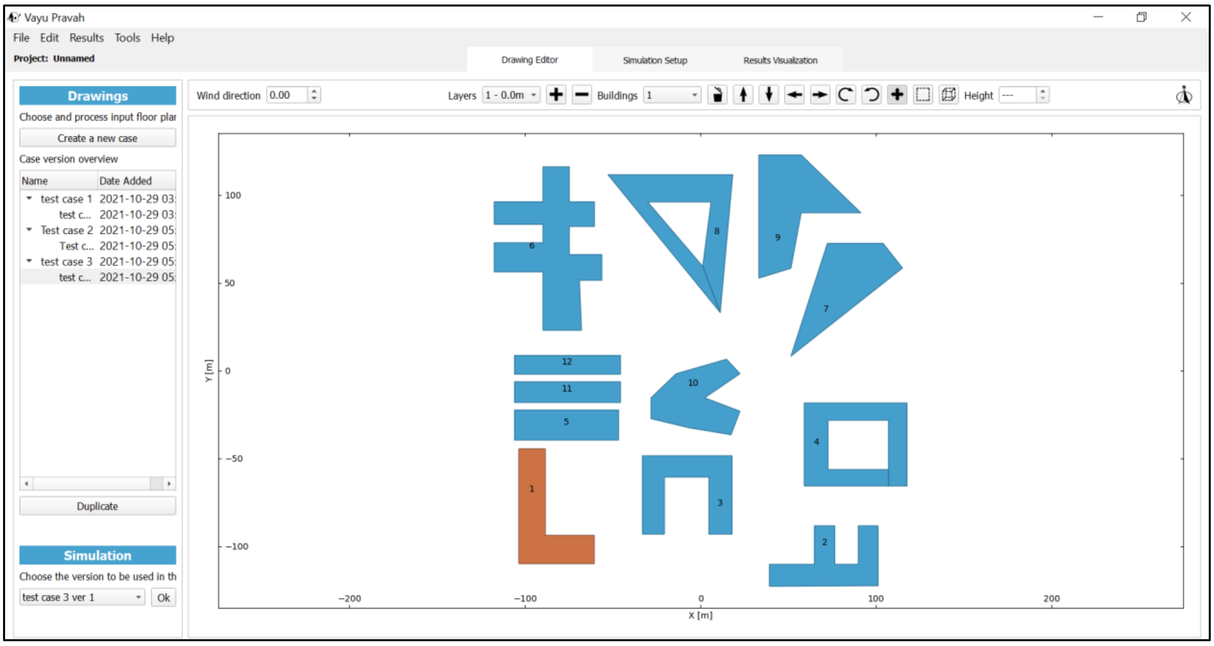


Figure 3-20 Case version overview, (on the right-hand side)

3.2.10.2 Modification of the building massing and setting up wind direction.

Wind direction can be adjusted by adding the degree of rotation clockwise from North (toward East) at the top left of the drawing editor.

3.2.10.3 Setting the orientation of buildings

One can rotate and/or move the buildings in the editor by selecting the building number to be moved and using different arrows to make the required move.

For example, building n°1 in the the drawing below has been moved and rotated here.

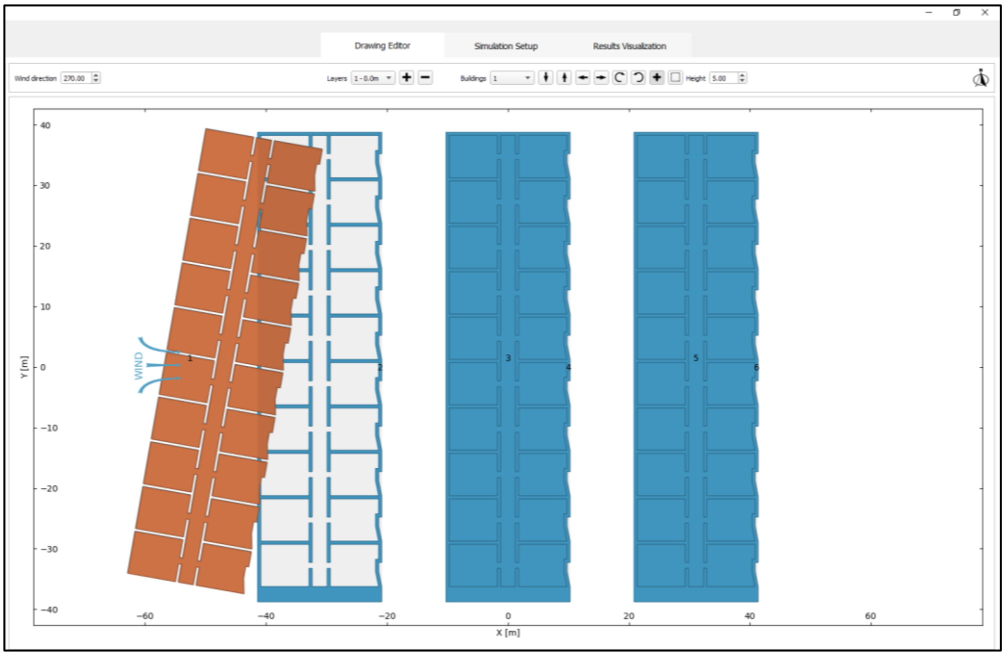


Figure 3-21 Building 1 (highlighted) is moved and rotated

The modification is saved and stored as a separate case in the project.

3.2.10.4 Preparing for the simulation setup

The case for simulation can be selected from the drop-down menu “Choose the case for selection,” in the right-hand pane and clicking Ok on the selected case.

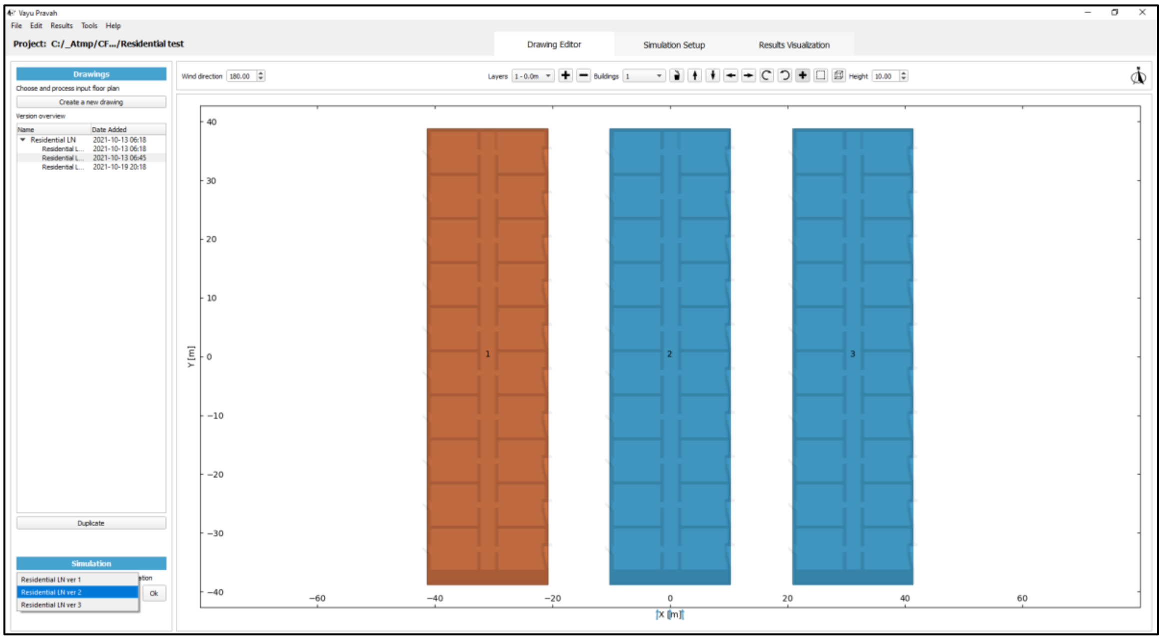


Figure 3-22 Selecting drawing version for simulation

3.3 Simulation Setup

Choosing a case for simulation and pressing “OK” leads the user to the simulation setup screen. Here the user can prepare the simulation case by entering building parameters such as height, wind direction, reference height, surroundings etc.

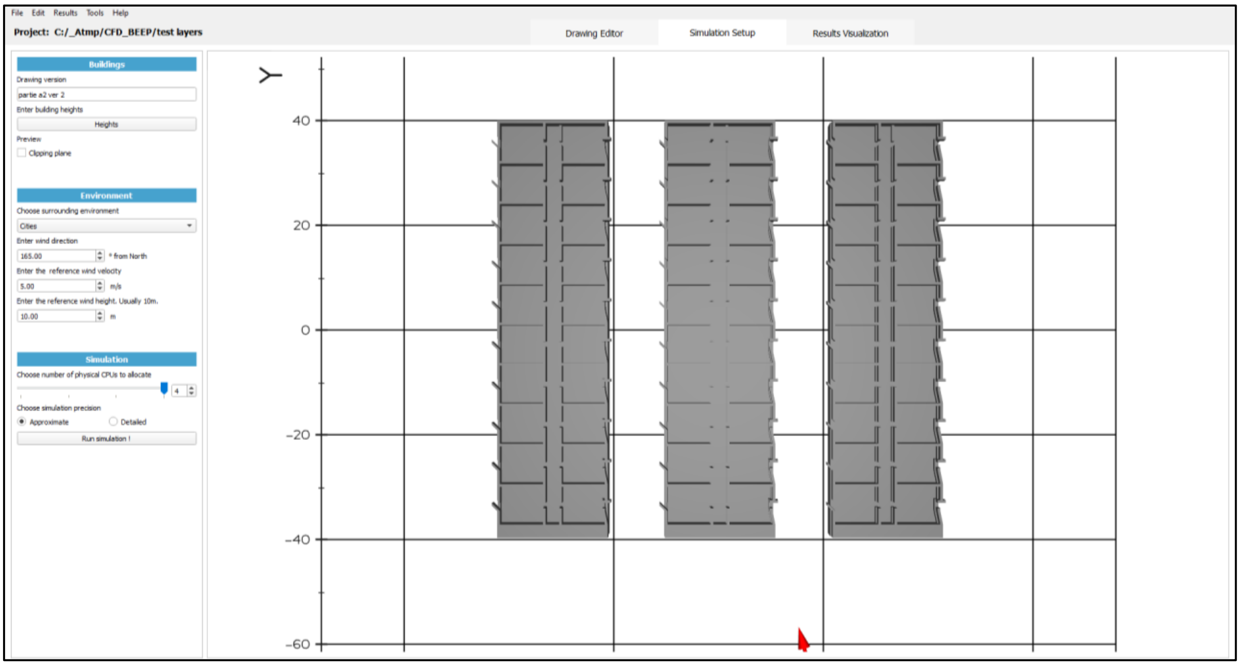


Figure 3-23 Simulation setup

3.3.1 Entering the buildings height

The user can enter heights in the simulation setup by clicking on 'Heights'. The buildings are numbered with their corresponding Layer number for better clarity.

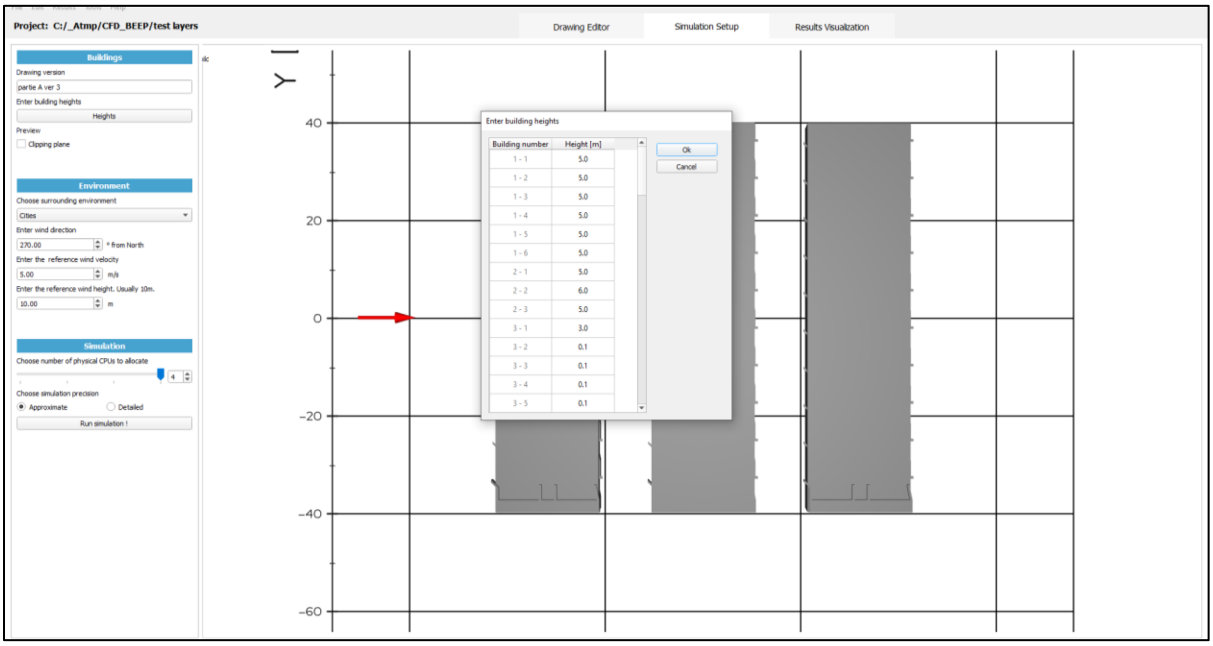


Figure 3-24 Building numbering is done for each layer separately for better clarity

Once this is done, one can proceed further by **clicking on the Ok button.** This will generate the 3 D model of the drawing basis the heights entered.

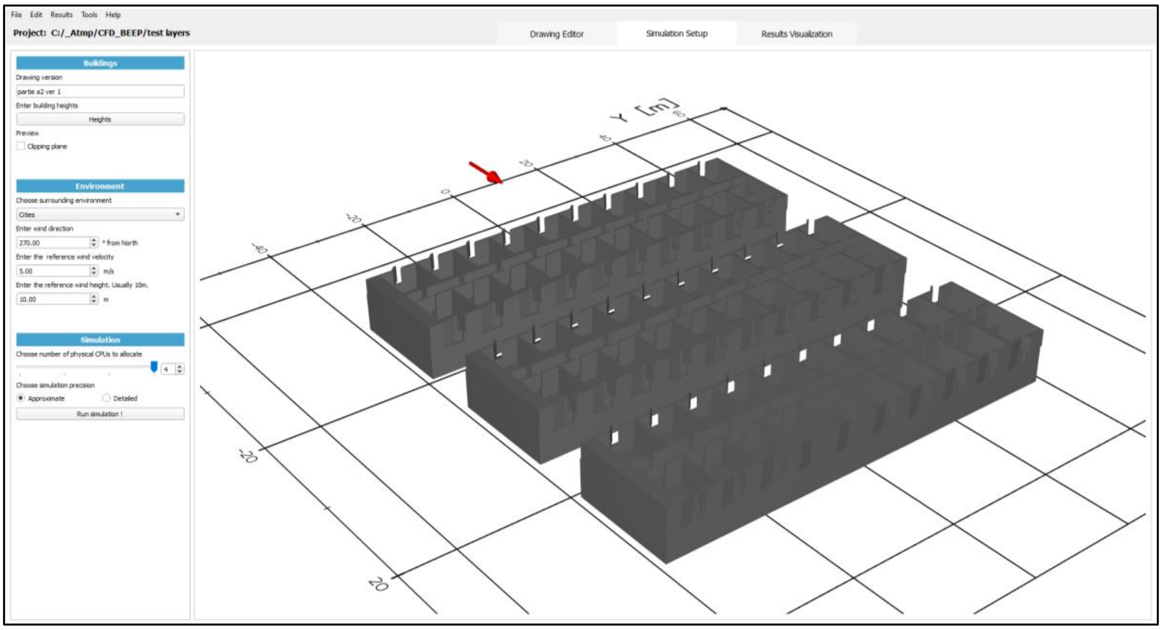


Figure 3-25 3-D model generated

3.3.2 Clipping pane

The user can click on the 'Clipping Plane' to clip the 3 D view along a specified axis.

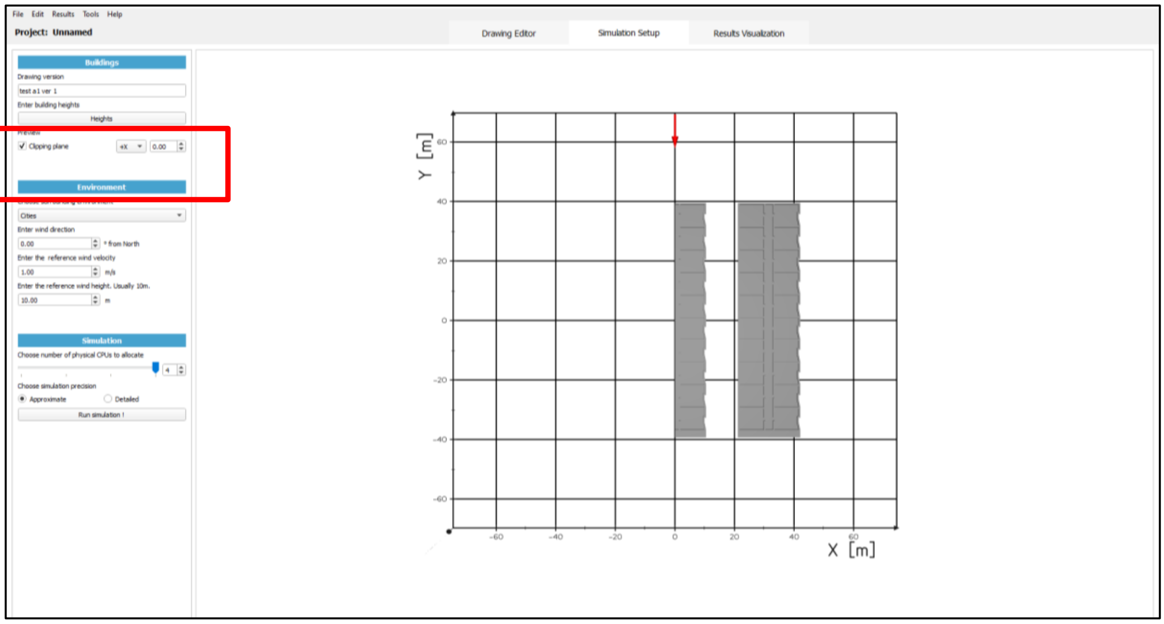


Figure 3-26 Clipping the view along an axis

3.3.3 Prepare the simulation

The drawing is then prepared for simulation by defining the environment and the wind conditions for the simulation.

3.3.3.1 Choose surrounding environment

The surrounding of the building massing (wind profile typology) can be chosen from the drop-down menu (cities, village etc.)

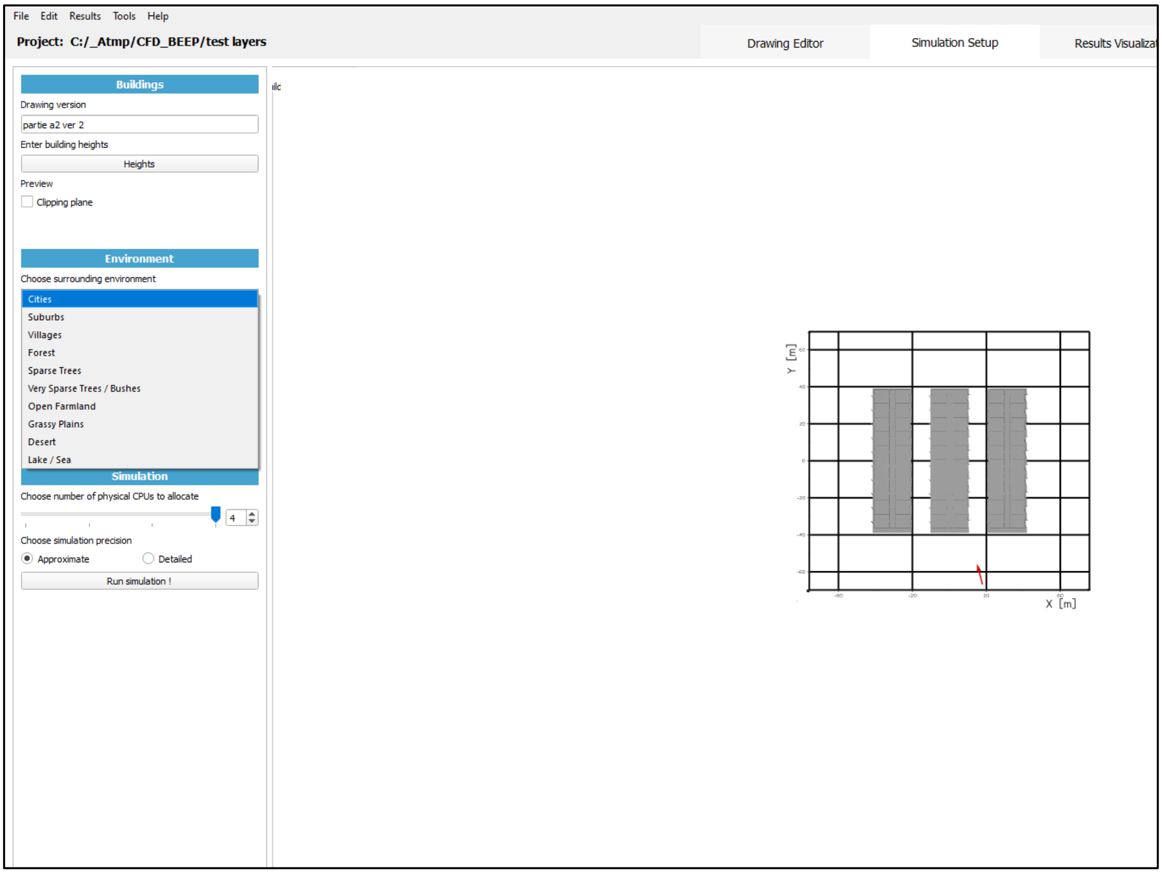


Figure 3-27 Selecting the surrounding environment for simulation

3.3.3.2 Choose the wind profile

Wind profile can be defined by choosing the wind direction and entering reference wind velocity and reference wind height. This will be reflected in the 3D model.

3.3.3.3 Chose the number of CPU to allocate to the simulation

Normally, significant reduction in simulation duration is obtained by choosing up to 4 physical cores (if available on the machine).

3.3.3.4 Chose the simulation precision model

The user can choose to have either an approximate or detailed simulation. During work meetings, the approximate mode can be used to help understand tendencies. The approximate version takes a few minutes to obtain a result indicating the main tendencies.

The Detailed simulation may take between 1 and 3 hours or more depending on the building size, numbers, and geometry, and the computer performance.

Before running the simulation, the project should be saved with a name, details (optional) and specifying a saving location. The project is now ready to be run for simulation.

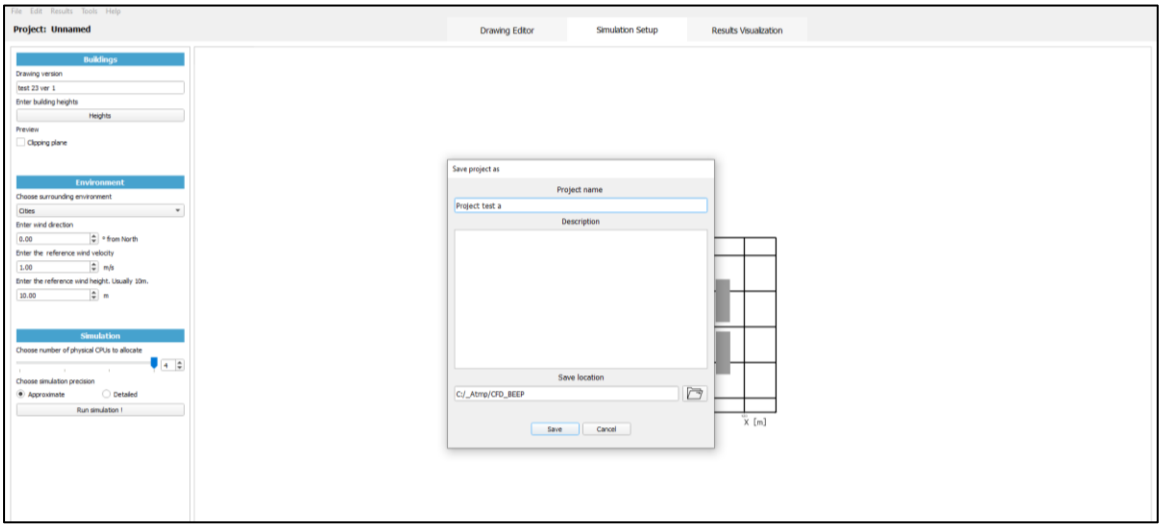


Figure 3-28 Saving a file by specifying name, description, and location

3.3.4 During the simulation

A pop-up window will appear indicating the status of the simulation. A first bar indicates that the meshing is ongoing. Once it is finished, the simulation blue band shows the progress and the estimation of the remaining time before the end of the simulation.

3.3.4.1 Simulation running

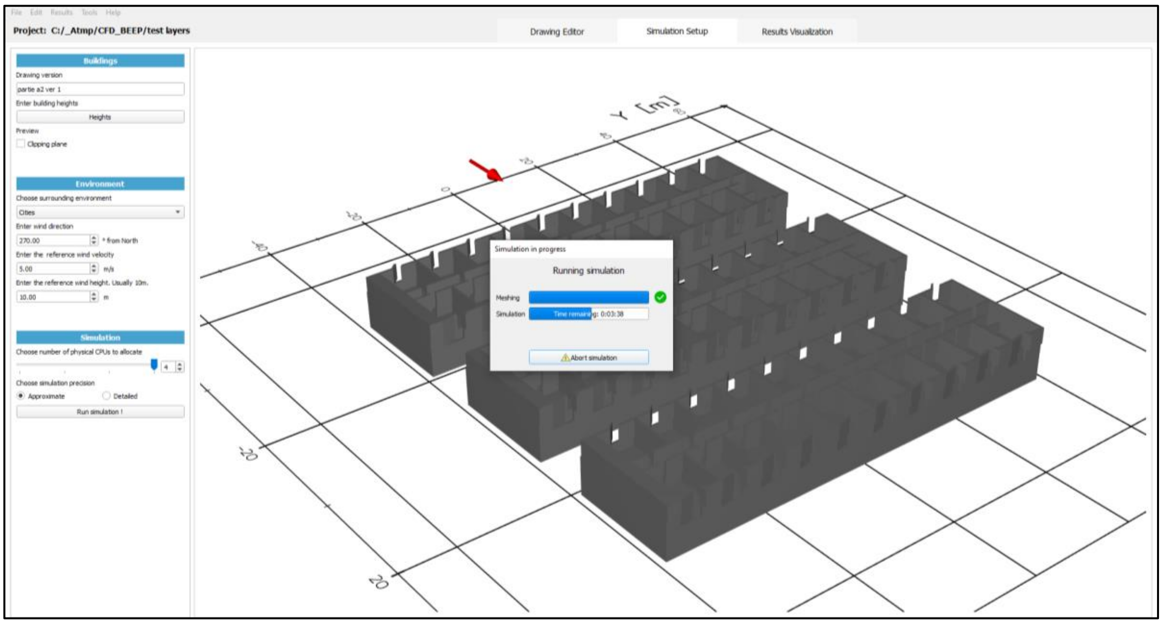
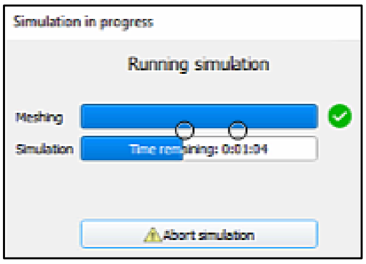


Figure 3-29 Simulation running with estimation of remaining time (after meshing)



At any stage, the simulation can be cancelled or 'aborted' by clicking on Abort simulation.

3.3.4.2 The simulation is completed

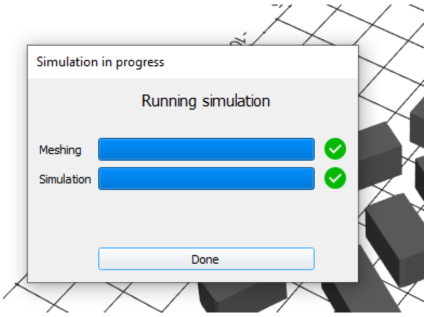


Figure 3-30 Simulation completed

3.4 Results Visualization

Once the simulation is complete, the user can move to the Results Visualization tab to view and analyze the simulation results.

3.4.1 Select the Results Visualization pane

Left click with the mouse on the top banner on Results Visualization to enter the visualization pane.

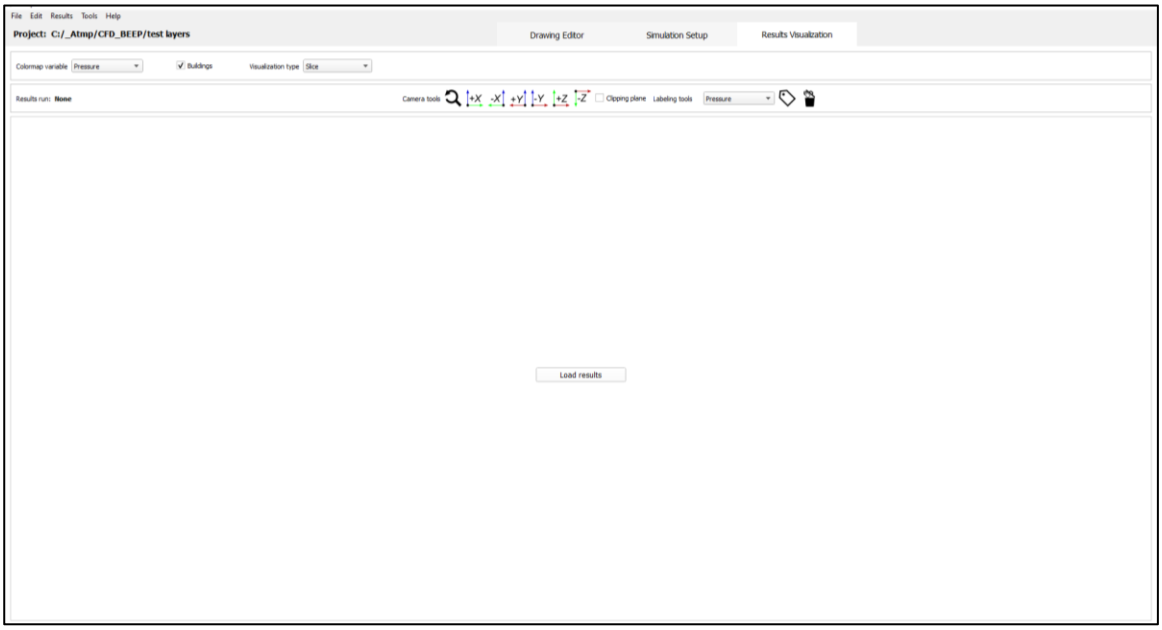


Figure 3-31 The Results Visualisation pane

3.4.1.1 Loading results

To view the results of the simulation, left click on the 'Load results' button.

3.4.1.2 Select the simulation just completed

Left click on the ok button and select the case for which simulation results are to be viewed. The 'reload results' window is in a matrix format where each simulation is shown along with the related simulation parameters, which leads to clear identification and selection of results to load.

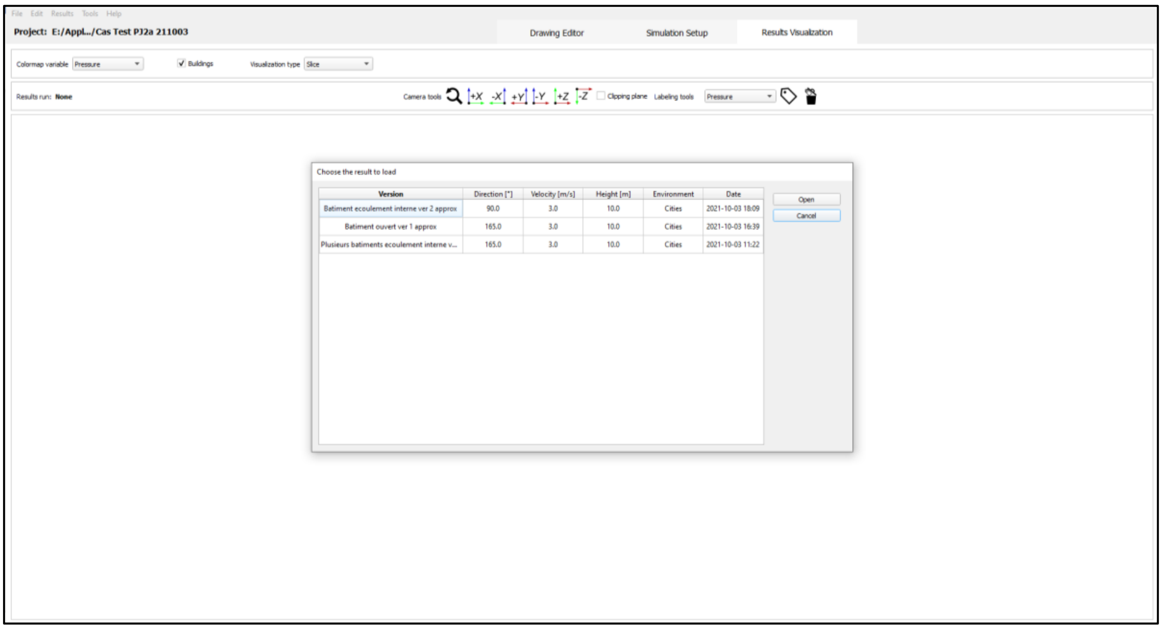


Figure 3-32 Selecting the case for which simulation results are to be viewed

3.4.2 Selecting, preparing, and adjusting the View

3.4.2.1 The different options of variables

One can select five different parameters to visualize in the simulation, along with its color on the result from the drop-down menu in the top left of the pane. These parameters are:

1) Pressure
2) Velocity magnitude (velocity direction as the resultant of all X, Y, Z components in the 3D space)
3) X-Velocity (component of the velocity in the X direction (wind direction in the results))
4) Y-Velocity (component of the velocity in the Y direction (horizontal transversal to the wind direction))
5) Z-Velocity (component of the velocity in the Z direction (vertical wind direction))

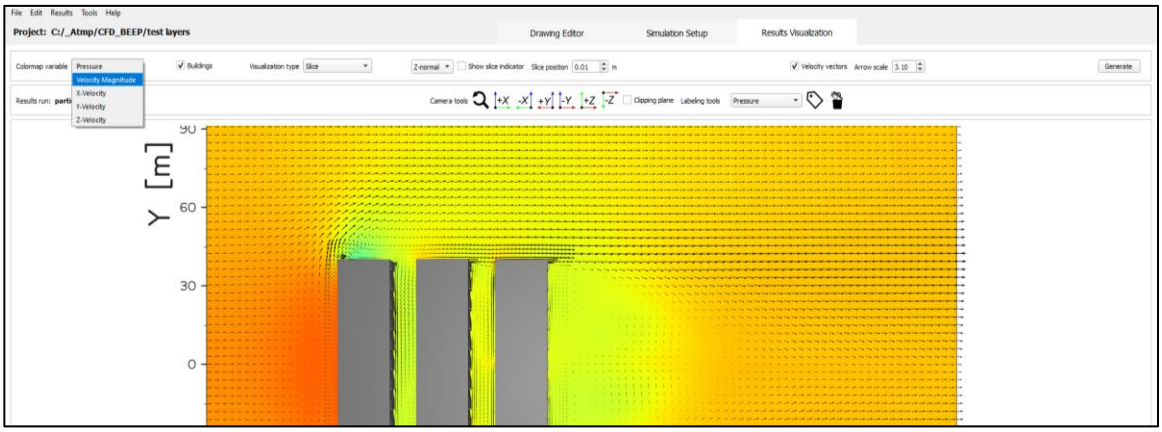


Figure 3-33 Selecting the variable to analyze in simulation results

3.4.2.2 Buildings in the 3D representation

The user can choose to either view or not, the buildings in the results by checking (or un-checking) the box "Buildings" on the top of the results screen.

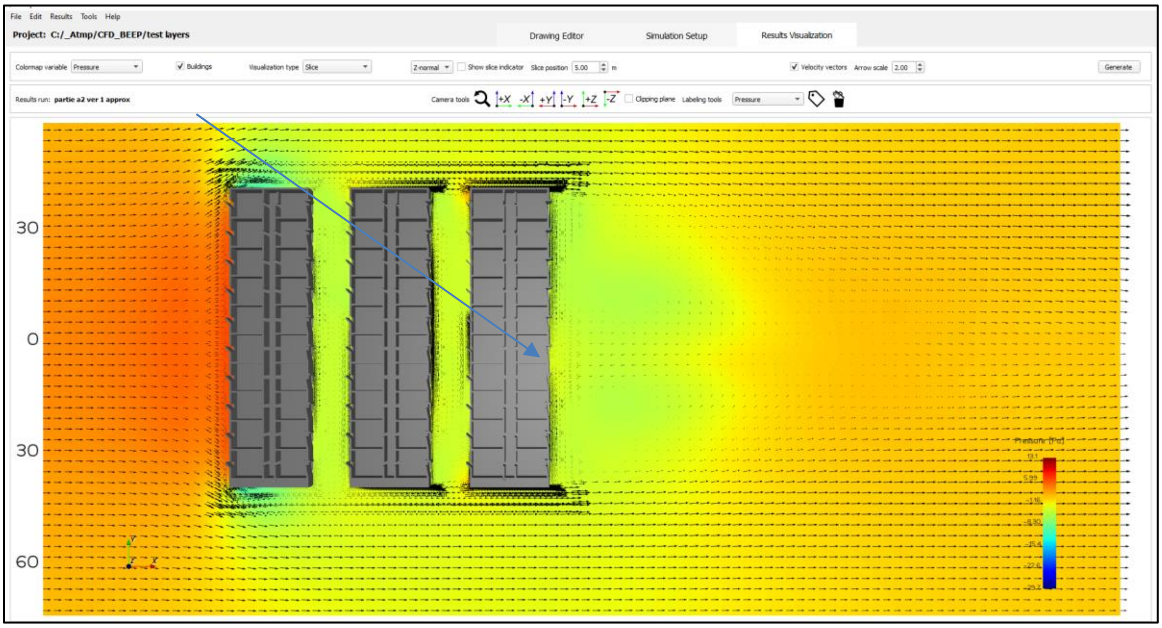


Figure 3-34 Simulation results with buildings

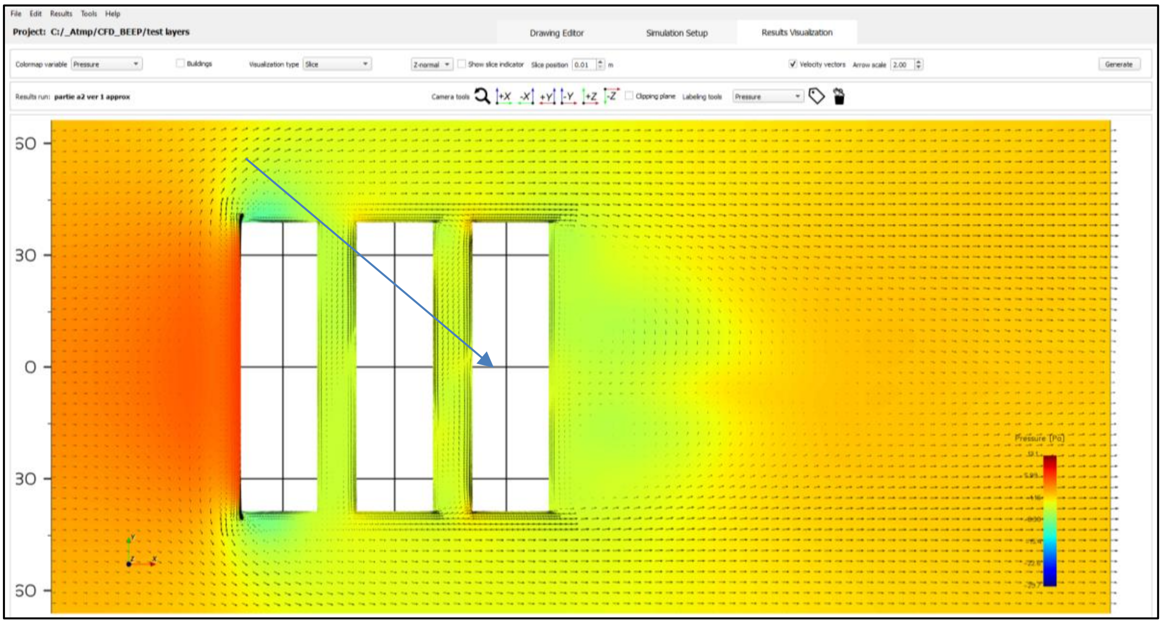
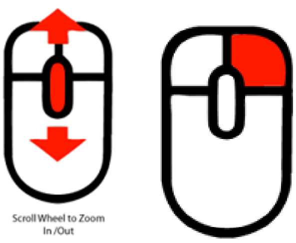


Figure 3-35 Simulation result without buildings shown

3.4.2.3 Zoom and un-zoom

Zooming is obtained by scrolling the center mouse wheel towards the front and un-zooming by rolling backwards.

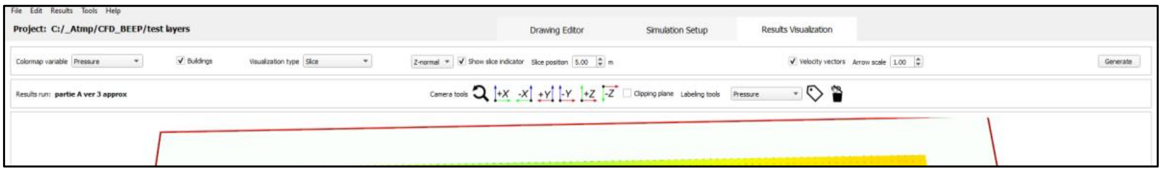


Zooming is obtained also by moving the mouse with the right button kept pushed (by sliding the mouse toward the screen)

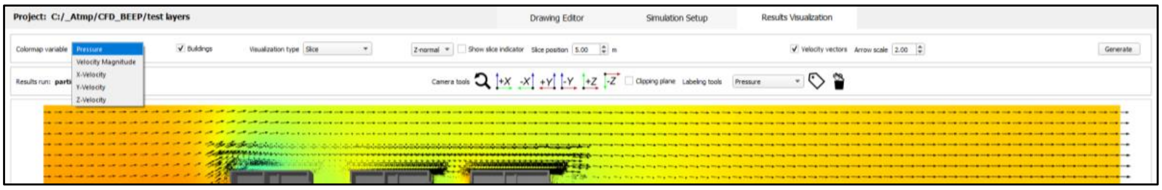
3.4.2.4 Panning with the cursor

Panning is performed by pushing the wheel of the mouse at the center and moving the mouse in the direction of the desired movement.

If the Show slice indicator is selected (ticked), then one can only pan if the cursor of the mouse is outside of the slice indicator rectangle.



If the Show slice indicator is not selected (unticked), then one can pan with the cursor in any position on the screen.



3.4.3 Selecting the kind of results presentation

One can select five different kinds of representation of the results for visualization.

3.4.3.1 Slice

This can be selected in the visualization drop down menu (as shown in figure). The slice visualization are planes perpendicular to the X, Y, or Z axes which can show the colored 5 variables (The different options of variables as mentioned in earlier), this example show the colored pressure values in horizontal plane (Z-normal, perpendicular to the vertical axis) at a height of 5 m.

The ‘Show Slice’ indicator can be ticked on or off according to visualisation requirement.

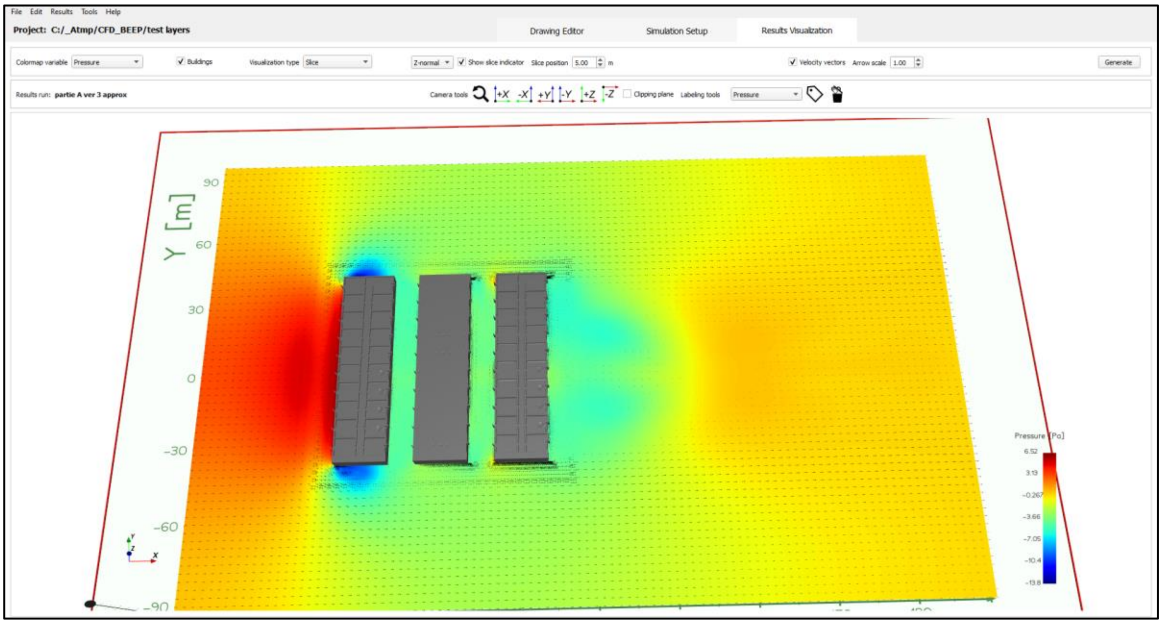


Figure 3-36 Simulation slice result with slice indicator shown

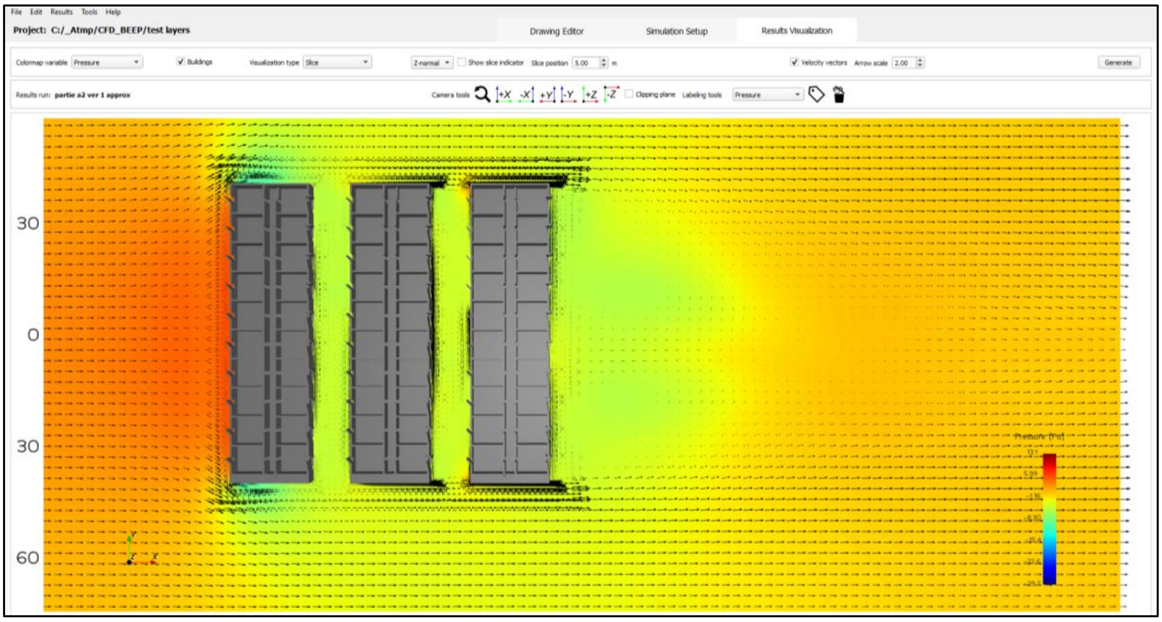


Figure 3-37 Slice visualization without slice indicator

3.4.3.1.1 Velocity vectors

In the slice visualization mode, one can show the velocity vectors by checking velocity vectors box and adjust their scale over a range (see figure above).

3.4.3.2 Threshold

This representation shows the volume in which the value of the threshold variable is in between two specified values. This visualization can be selected in the visualization type drop down menu and selecting the parameter on which the threshold is to be applied. The range of values can then be defined in the adjacent bar.

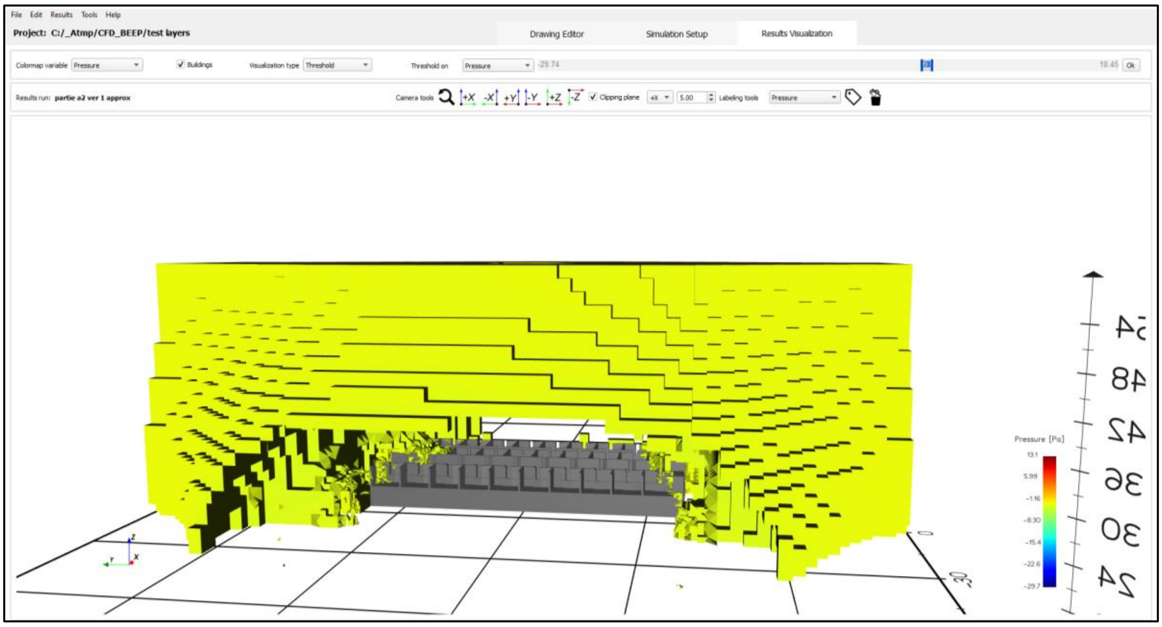


Figure 3-38 Threshold type visualization

3.4.3.3 Streamlines

A streamline is a line in the fluid whose tangent is everywhere parallel to the local velocity vector instantaneously. It shows the general pattern of air flow movement, is more informative than quantitative, but can help in explanations and presentations. The intensity and the domain size for visualization can be adjusted by the streamline density bar (from low to high) and clicking on 'Generate'.

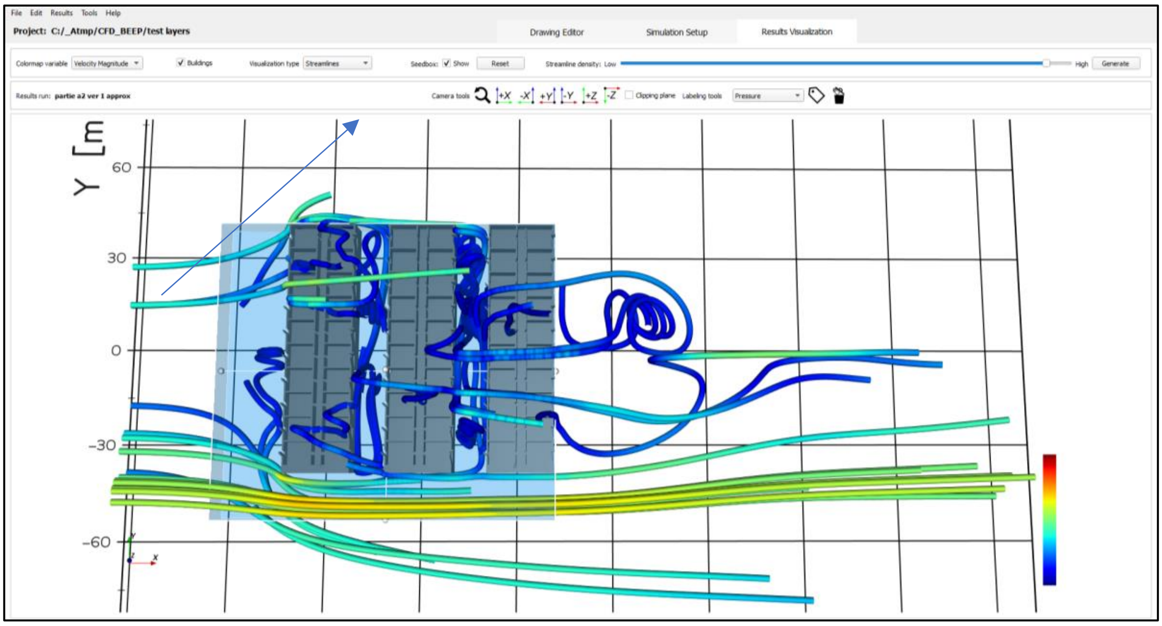


Figure 3-39 Streamline visualization

One can chose if the colors of the streamlines are set to the pressure or to the velocity.

One can also vary the density of the streamlines and adjust the size of the streamline box (seed box) on the three axes.

3.4.3.4 Building surfaces

This visualization displays the buildings with their façade colored by either one of the fives possible variables (pressure, velocity magnitude, X, Y or Z velocity component). They are used to determine the potential of the wind velocity and pressure on each façade. Pressure on the building surface can be used for simplified calculation for air change.

Figure 3-40 Building surface visualization, for viewing parameters at the building facade

3.4.3.5 Labels

Choosing the labels variables

Each of the visualization allows to add labels displaying values of the variables like the variables of the color maps by choosing Labelling tools and selecting the parameter from the drop-down menu.

1) Pressure
2) Velocity magnitude (velocity direction as the resultant of all X, Y, Z components in the 3D space)
3) X-Velocity (component of the velocity in the X direction (wind direction in the results)
4) Y-Velocity (component of the velocity in the Y direction (horizontal transversal to the wind direction)
5) Z-Velocity (component of the velocity in the Z direction (vertical wind direction)

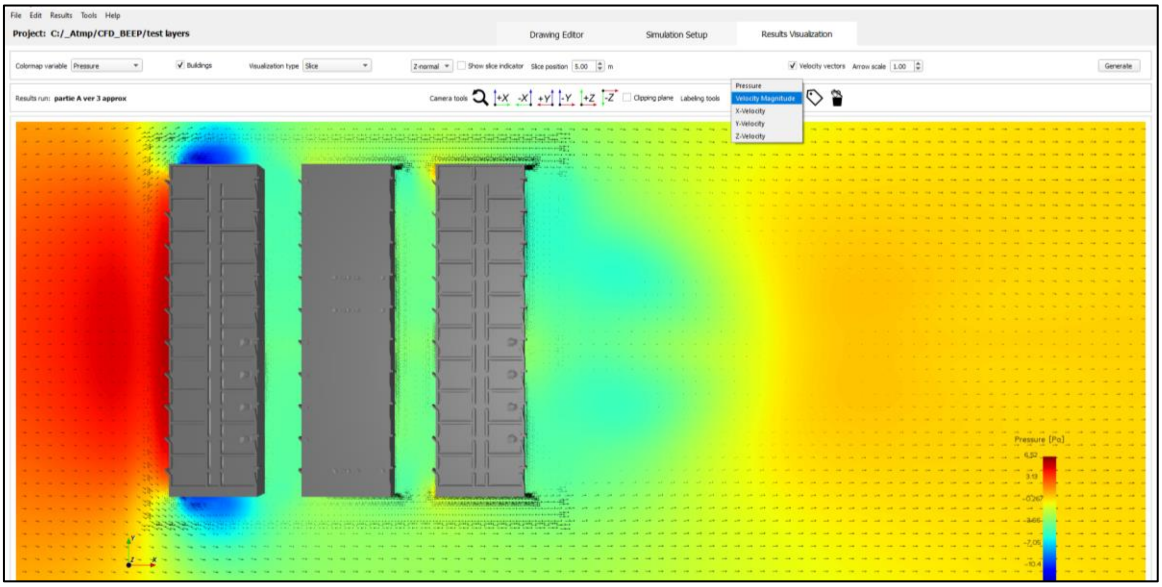


Figure 3-41 Inserting quantitative labels in the simulation result

Placing labels

Once the variable is chosen, the user can left click on the label icon and then to move the mouse cursor to the location of the labelling and left click.

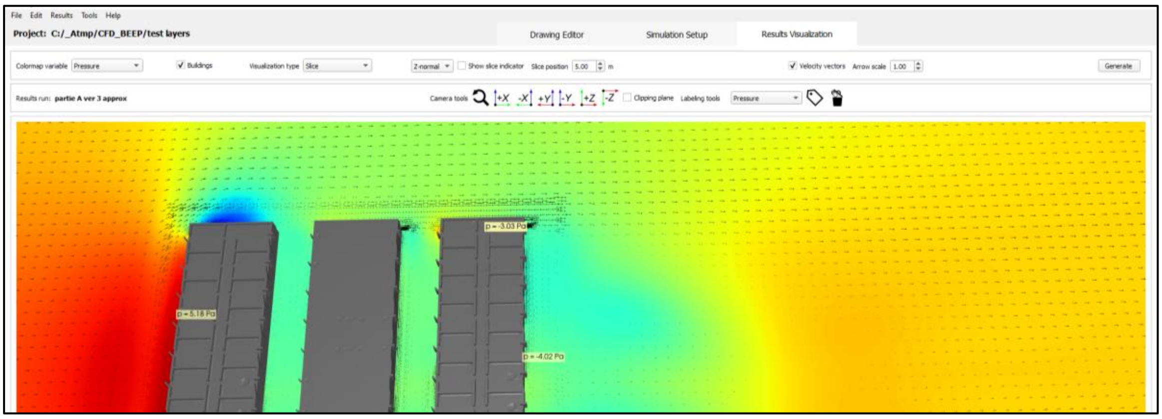


Figure 3-42 Results with pressure labels added

The labels are attached to one plan level, the value is fixed, if one changes the position of the visualization plane position, they disappear.

They can also be added on the building surface representation.

To remove the labels, press on the "" icon next to the Label icon.

The scales of the variable representation can be adjusted on the Results Visualization panel to customize the results view.

3.4.3.6 Adjusting the color map for the pressure, velocity magnitude and X, Y, Z directions of the velocity

Under the results menu, select the Use of Custom Colormap Range

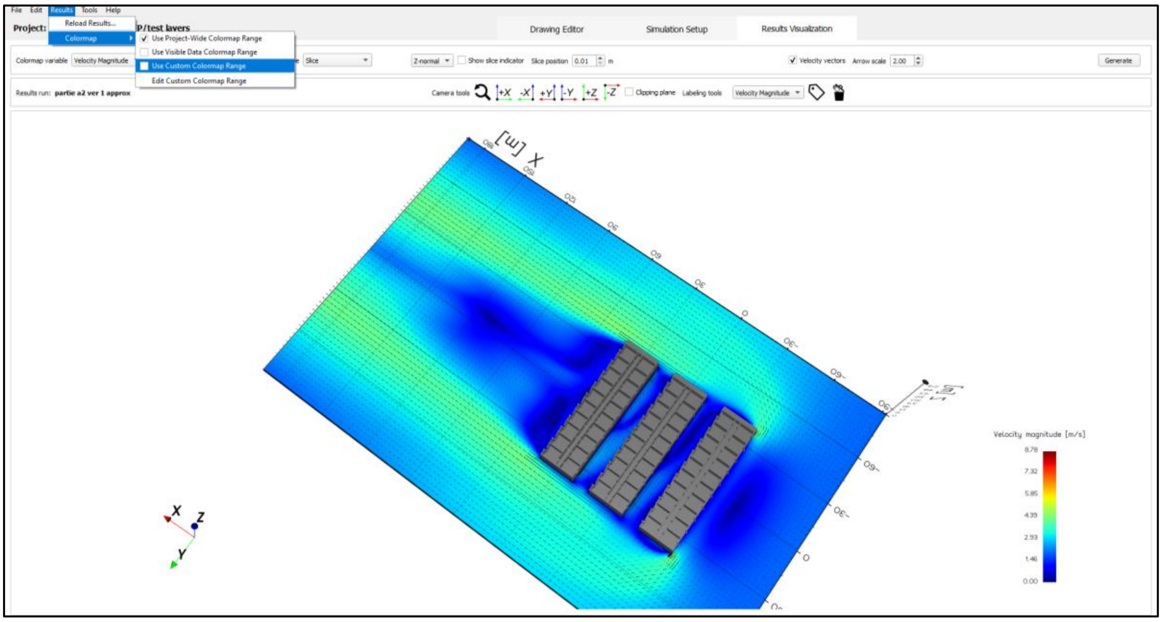


Figure 3-43 Adjusting color scale

Adjust the values for the variables Pressure, Velocity magnitude, X-velocity, Y-Velocity, Z-Velocity

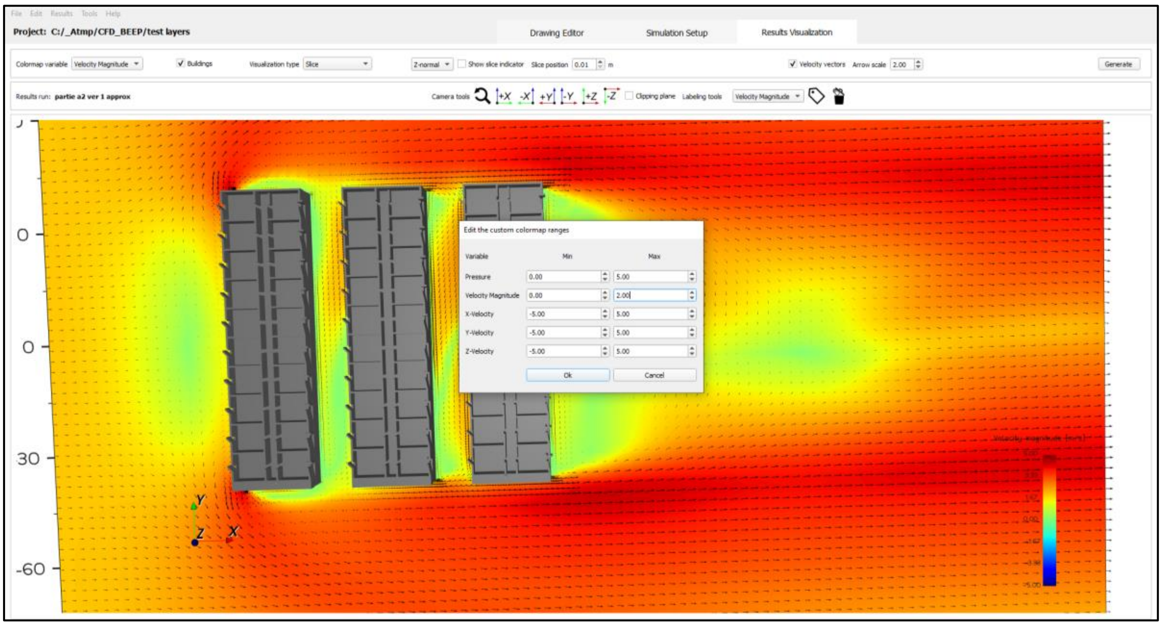


Figure 3-44 Adjusting the variable range

Click Ok to validate.

3.4.4 Visualization type

3.4.4.1 Pressure in a horizontal plane (Z-normal)

The graph below shows the pressure values (for colors see scale on the scale on the right-hand side) on a horizontal plane.

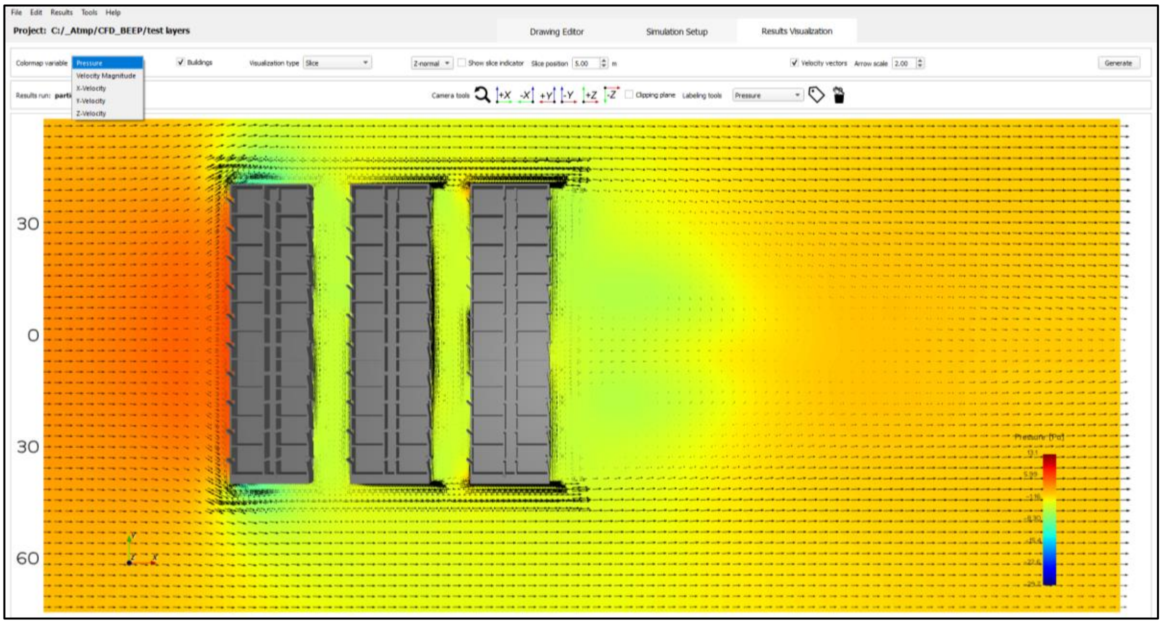


Figure 3-45 Visualizing pressure variable

In this visualization case, the user can choose to show the velocity vectors or not.

3.4.4.2 Axis denomination

The axis along the wind direction is the X axis, the axis perpendicular to the wind direction is the Y axis, the vertical axis is the Z axis. For example, a horizontal plane qualified as Z-normal, a plane perpendicular to the wind direction as X-normal, a plane parallel to the wind direction as Y-normal.

3.4.5 Clipping Plane

The user can clip the result along a particular axis at a desired height to see the internal flows in the building. This is done by clicking on the check box left of ‘Clipping plane’, choosing the axis along which to clip and specifying the clip height.

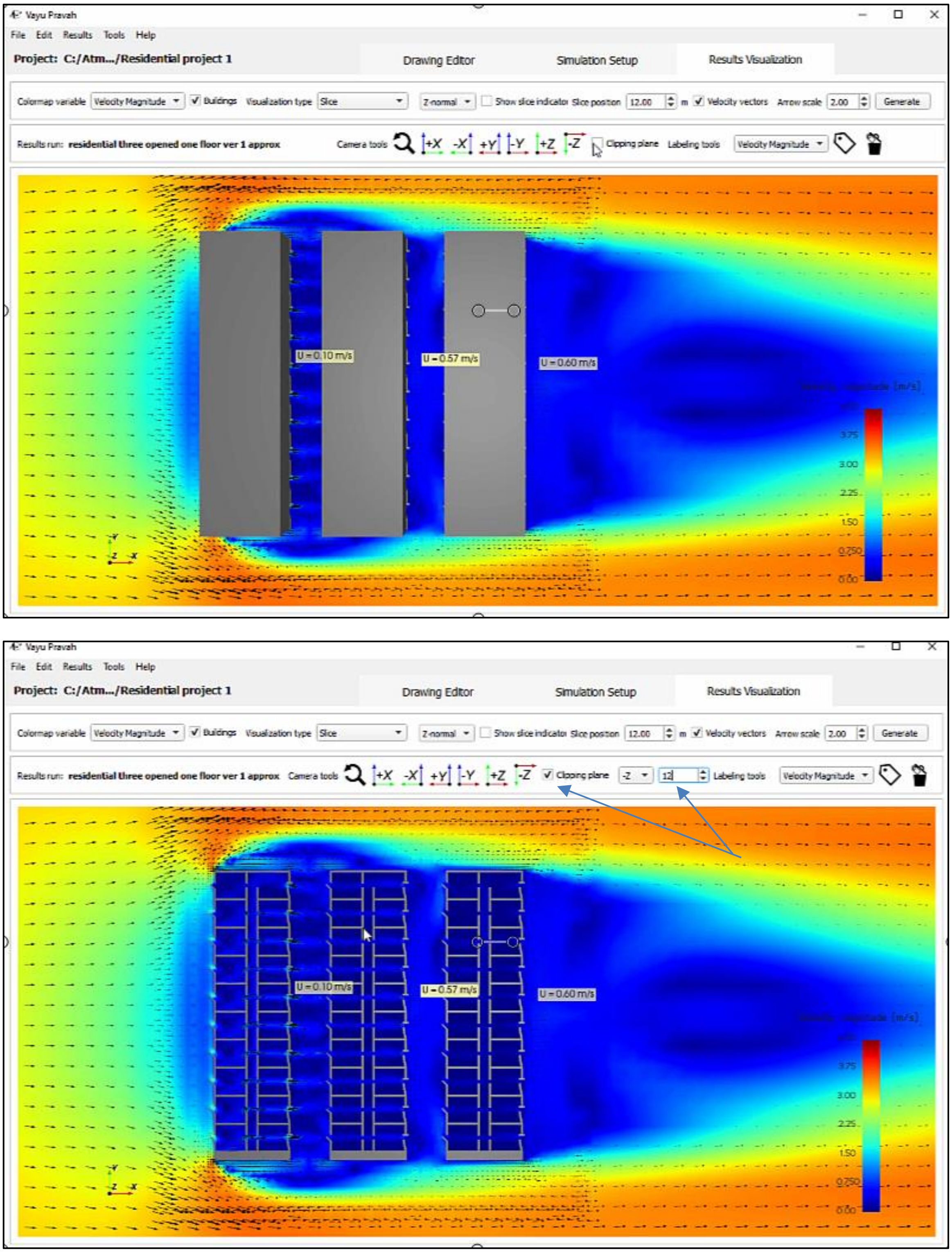


Figure 3-46 Without and with Clipped plane on Z axis

3.4.6 Generate pdfs reports

The program allows to automatically generate pdfs of slice mode plans at different intervals.

Under the File option, clicking on the Generate PDF report, the pop-up menu allows the user to select the colormap variable to be mapped (pressure, velocity etc.), the variable color scale bounds, the plane orientation, the axis bounds and the interval between the planes. The pdfs can be used to systematically compare versions after calculations.

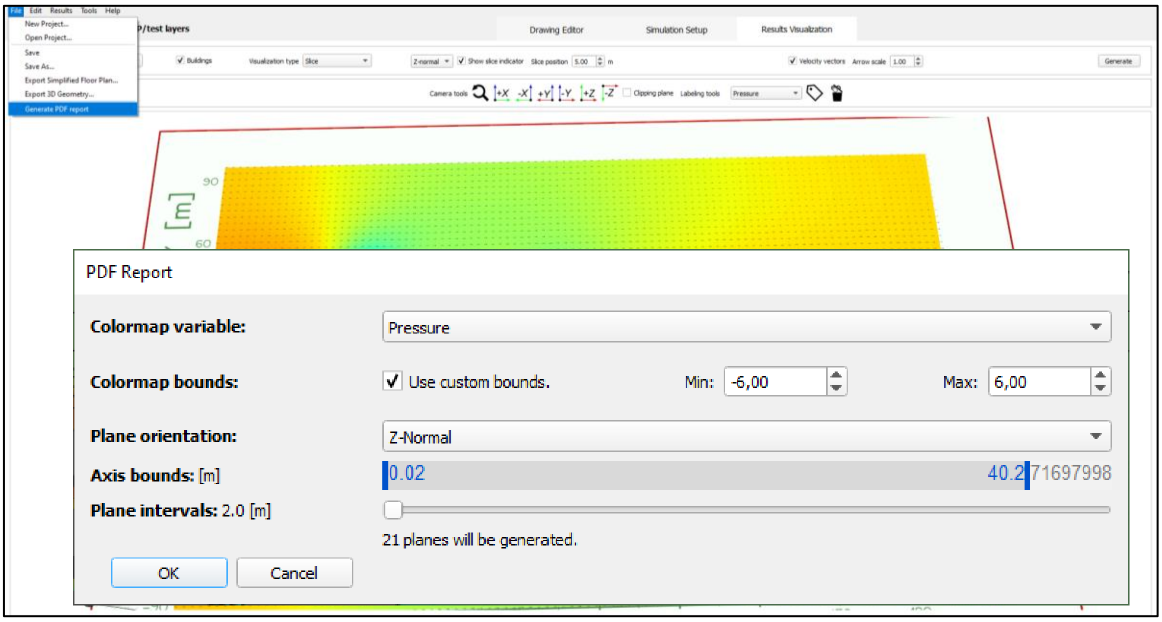


Figure 3-47 Preparing for a pdf report

Select the location of the report to be saved.

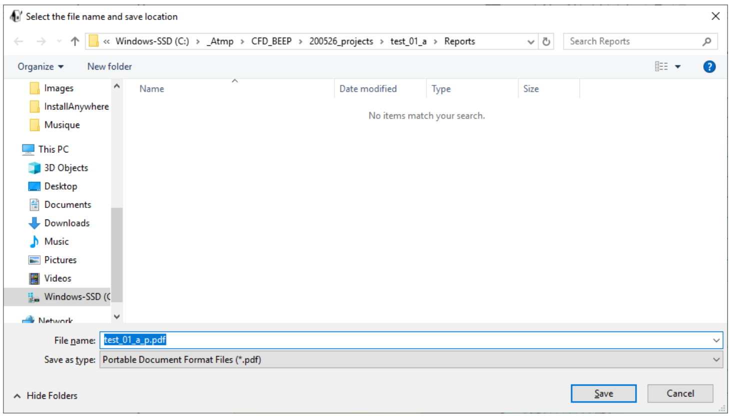


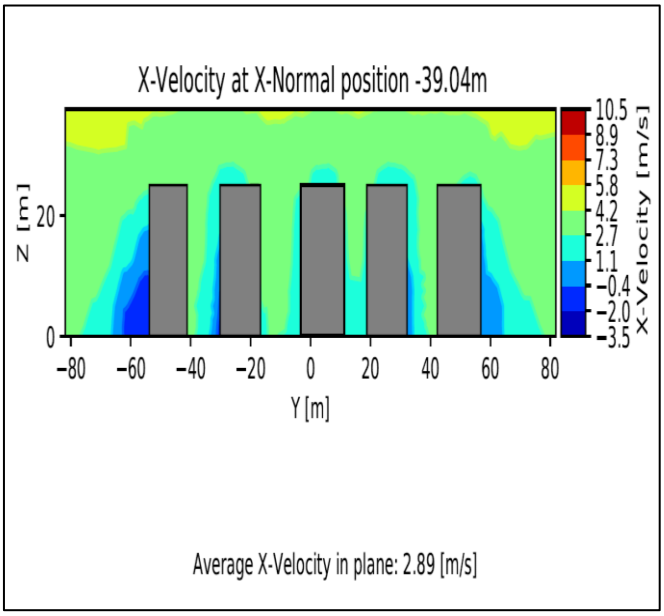
Figure 3-48 Saving the pdf report

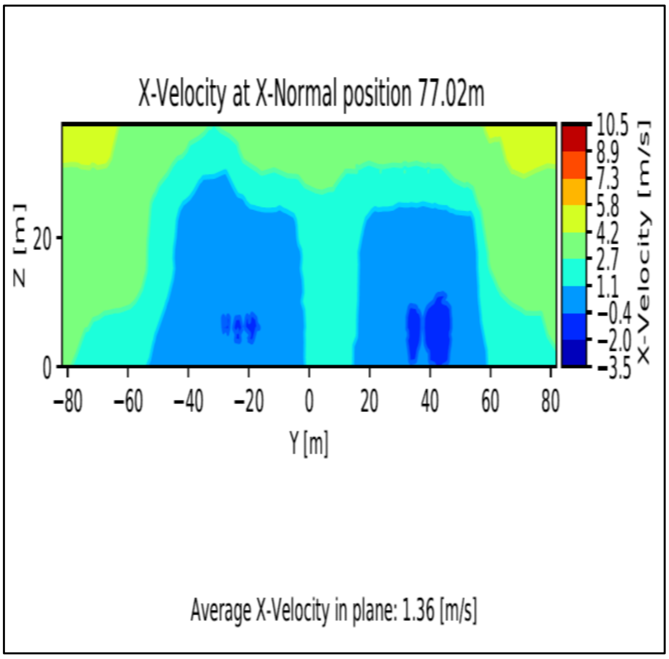
The file displays the planes with a zoom on the building domain.

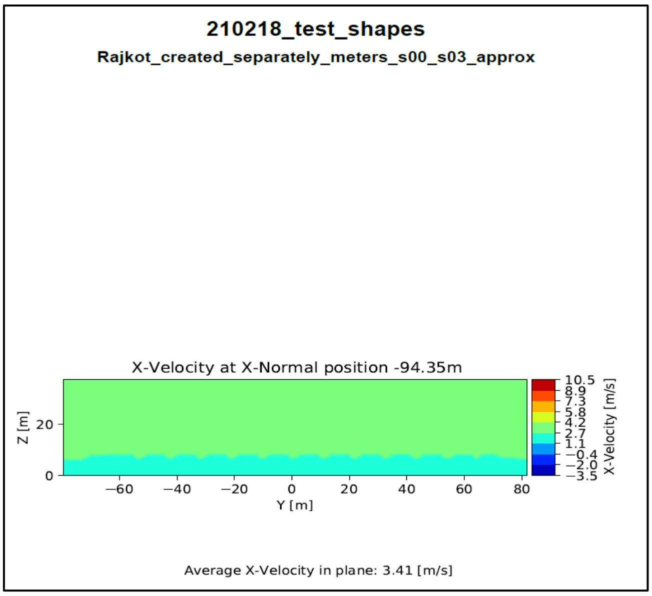
Example of analysis of Rajkot SMART GHAR buildings pdf reports

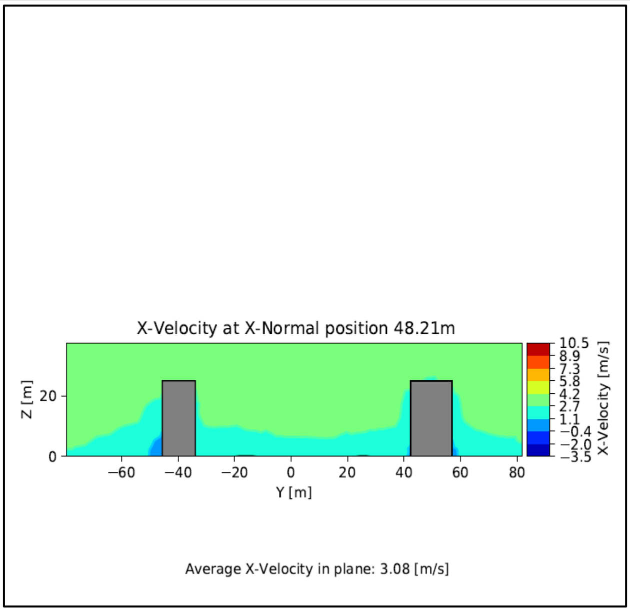
Analysis of the project design for a building can be done to make comparative study with different building heights, orientations etc. and the resulting wind flow.

Case 1: X velocity with standard building heights

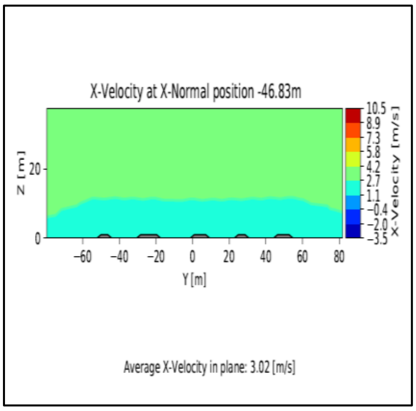

X-Velocity at X-Normal position -39.04m
Z [m]
20
0
−80
−60
−40
−20
0
20
40
60
80
Y [m]
10.5
8.9
7.3
5.8
4.2
2.7
1.1
−0.4
−2.0
−3.5
X-Velocity [m/s]
Average X-Velocity in plane: 2.89 [m/s]


X-Velocity at X-Normal position 77.02m
Z [m]
20
0
−80
−60
−40
−20
0
20
40
60
80
Y [m]
10.5
8.9
7.3
5.8
4.2
2.7
1.1
−0.4
−2.0
−3.5
X-Velocity [m/s]
Average X-Velocity in plane: 1.36 [m/s]

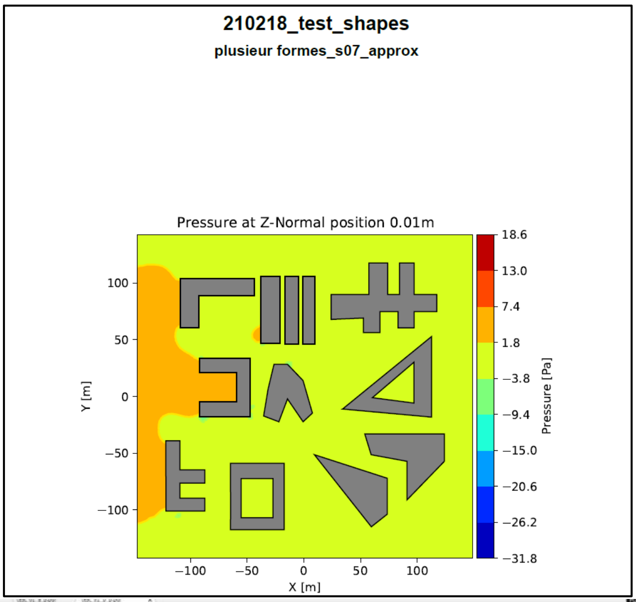


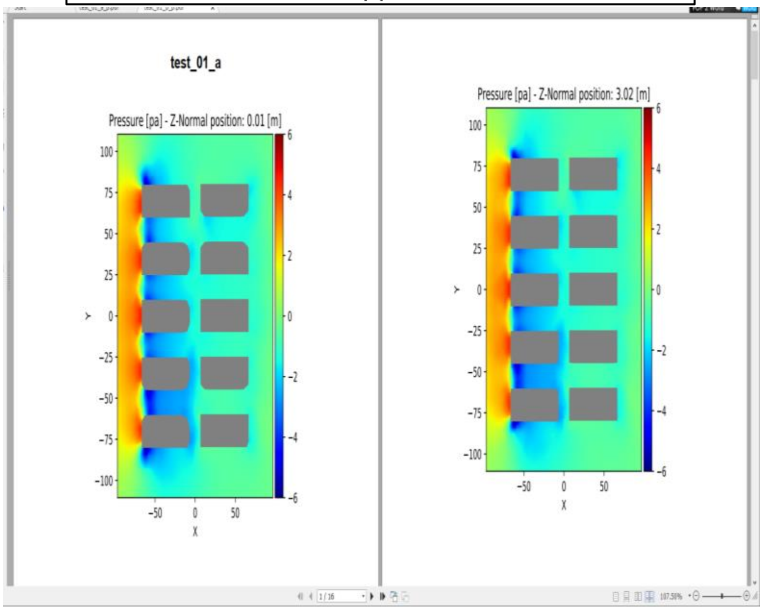


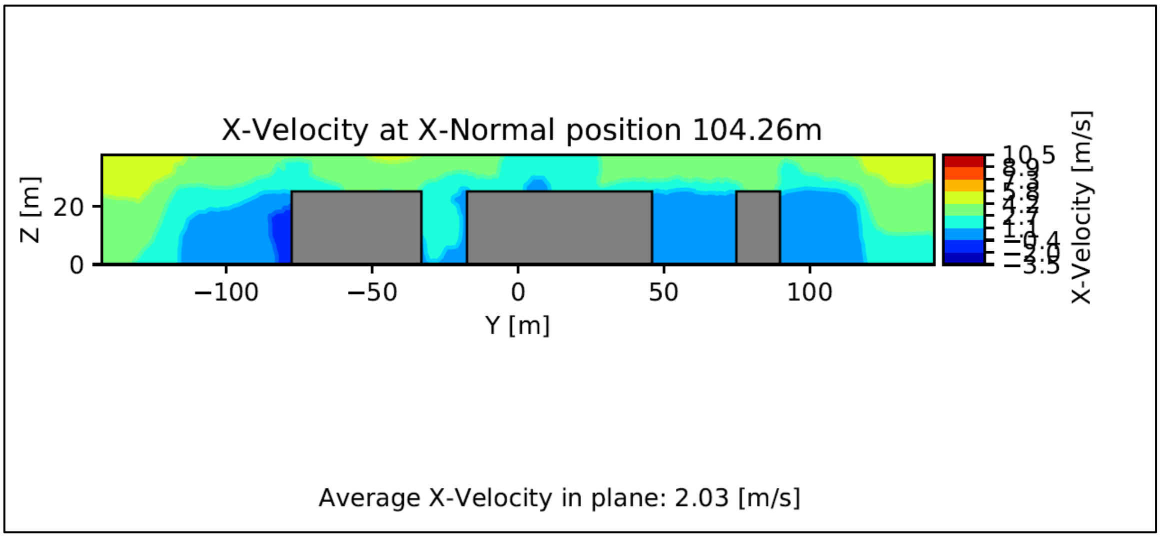
Case 2 : X velocity analysis with heights of all but two buildings minimized to 1 meter.



Other examples:







3.4.7 Loading another project

An existing project can be loaded into the software under the Open Project option under File by selecting the project in the directory.

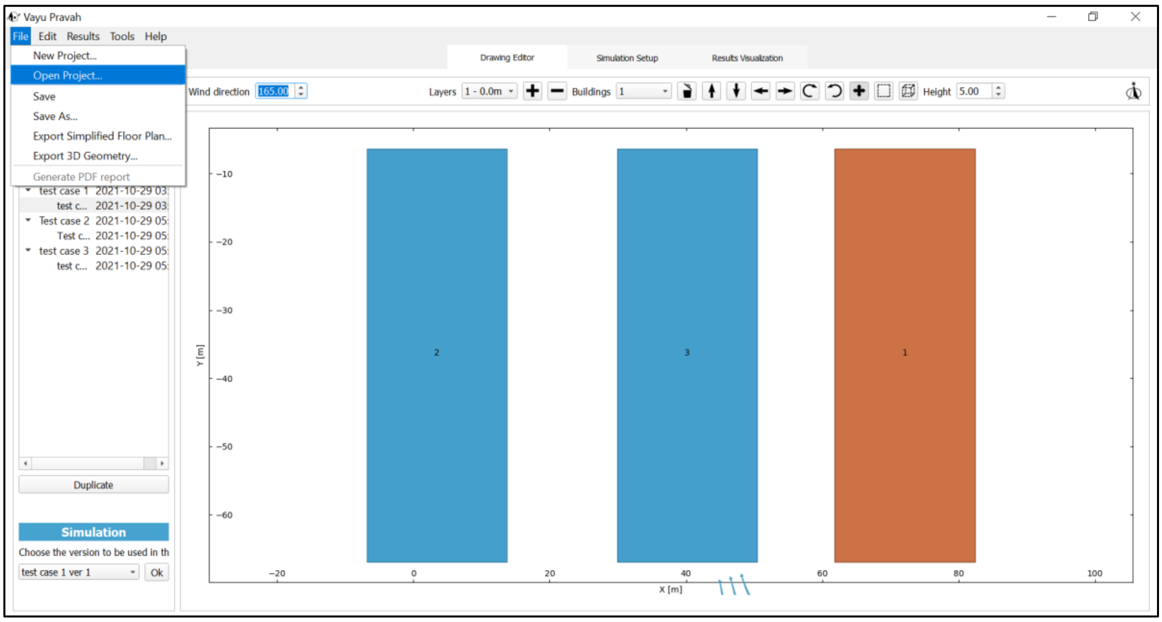


Figure 3-49 Opening an existing project

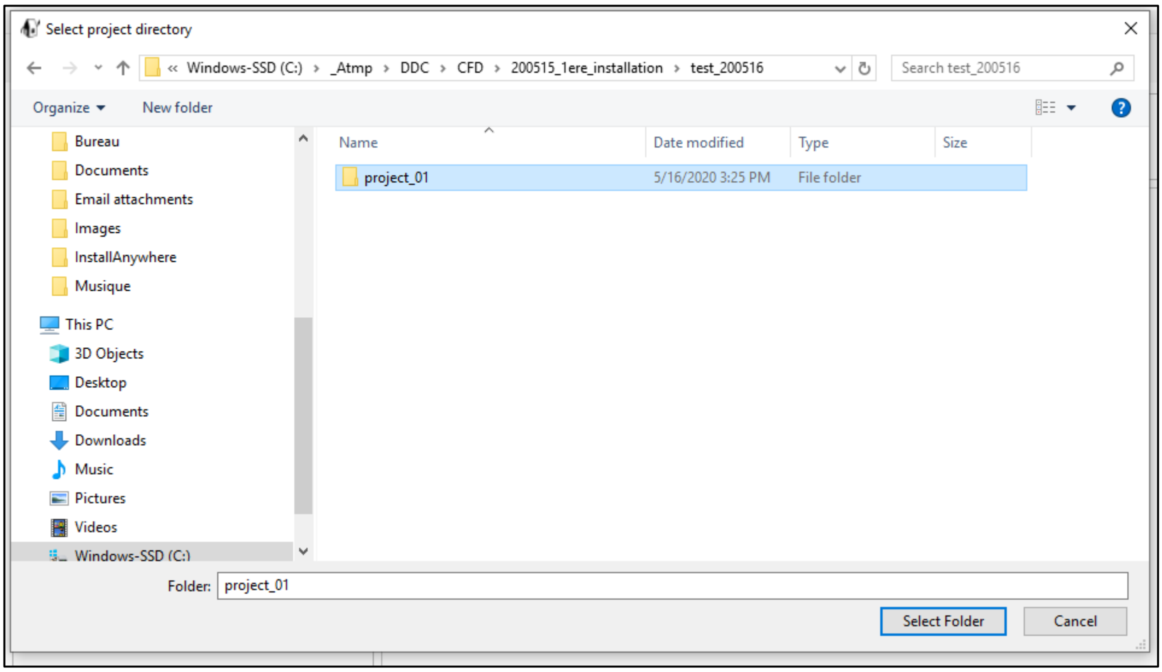


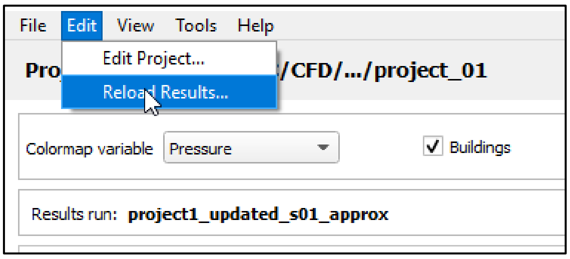
Figure 3-50 Choosing another project to be loaded

3.4.8 Load results from another case in the same project

The user can load results from another version of the same case or another case in a project (stored as a different case in the project) in the results pane. This is done using the reload results option. For each version one can generate pdfs with same parameters (range of pressure, velocity etc..) for comparing different cases.

3.4.8.1 Reload results

The reload results allows to load another option than the one shown.



3.4.8.2 Selecting one of the alternative designs

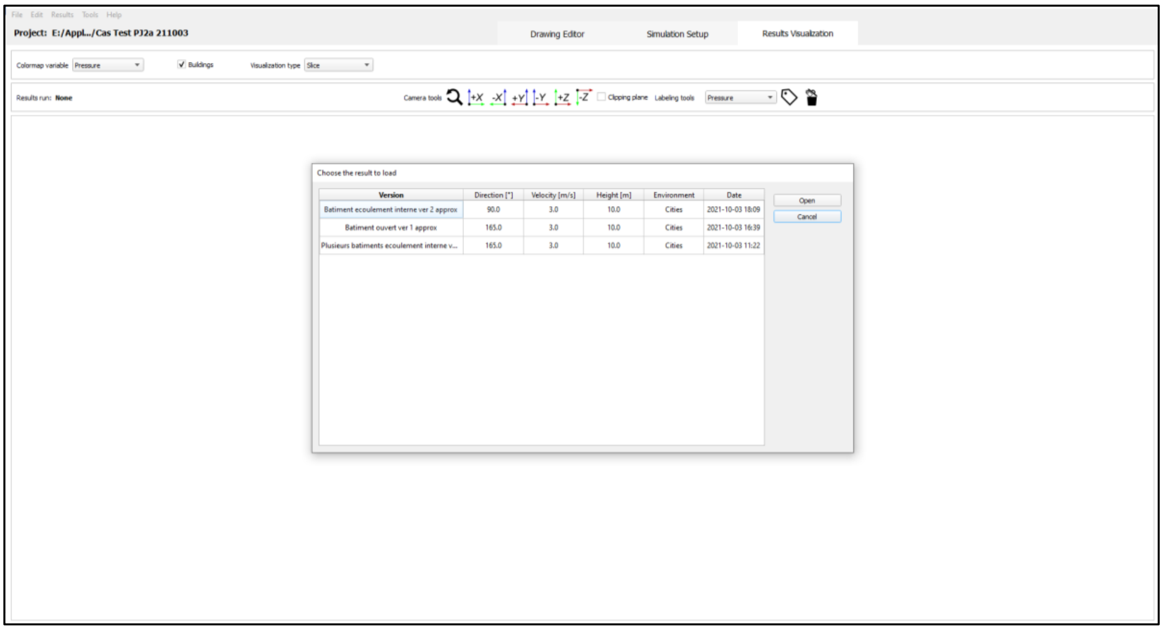


Figure 3-51 Choosing another simulation result to load

4 Buildings with external and internal air flows

This chapter introduces one important feature showing the possibility of having external and internal flows simulation. This chapter focuses on the different kinds of buildings with openings like fins and how the simulation is done in the Vayu Pravah software. Two such examples are being discussed here.

4.1 Closed building with fins

The figure shows a simple building design with fins. A design element as shown has been added for the meshing to indicate the closed building.

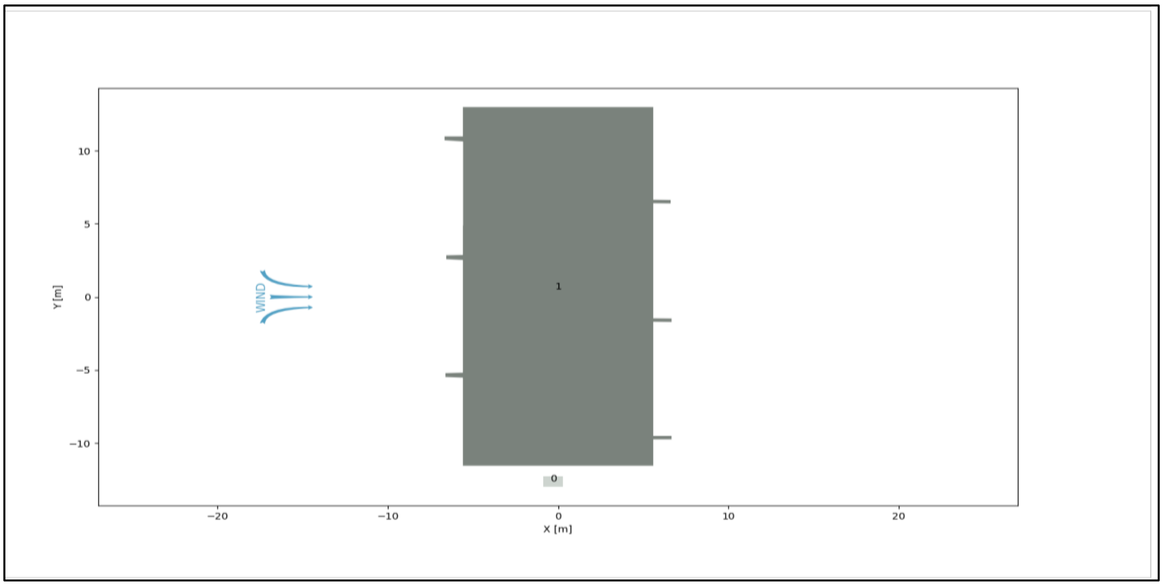


Figure 4-1 dxf file of building with fins

When the simulation is run for the same, we can see the results with the pressure labelled at the building façade.

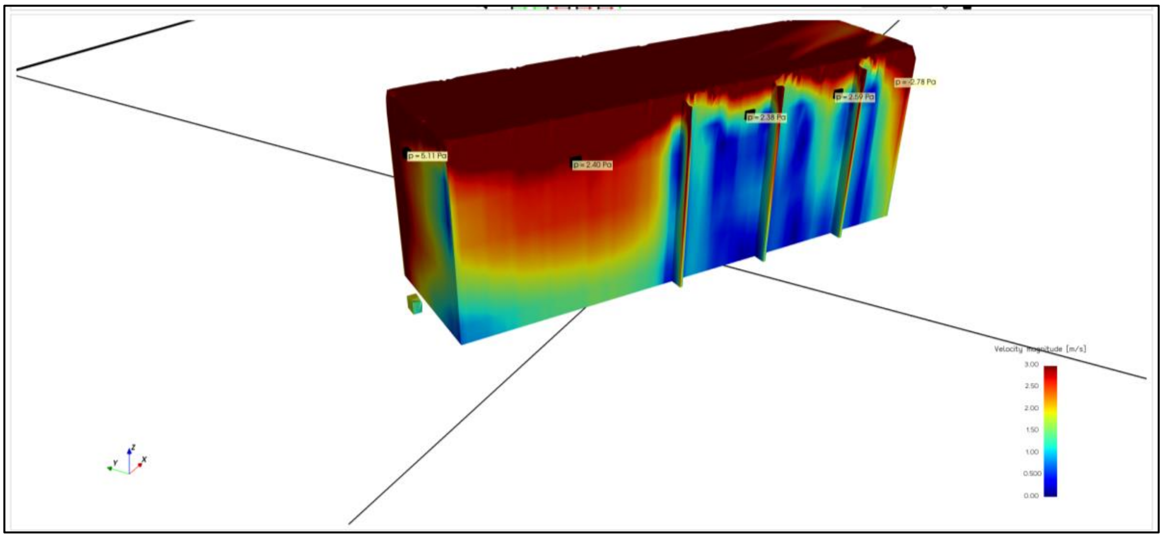


Figure 4-2 Simulation results for building with closed flow

4.2 Open building with fins

This is an example of a building with internal flows.

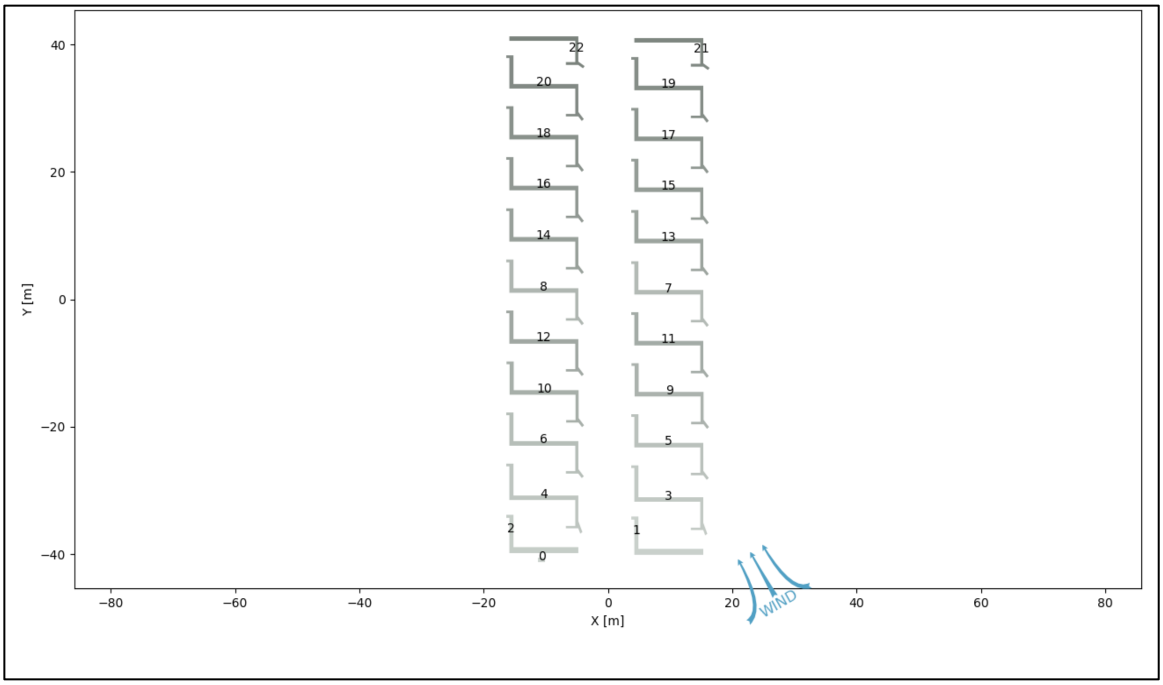


Figure 4-3 Dxf building with open flow

The simulation results of the building with internal flows is as shown below.

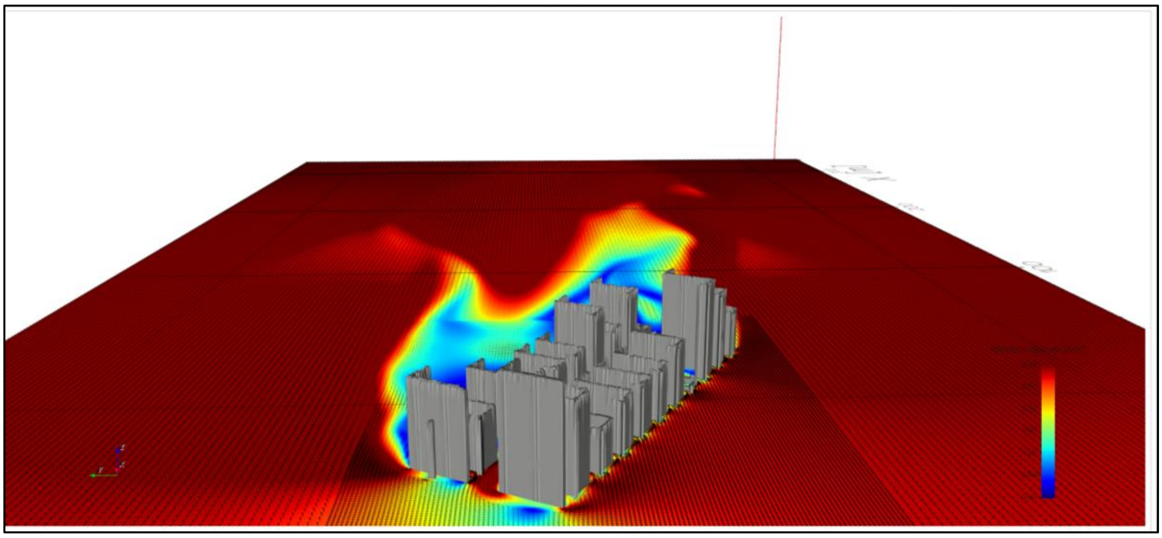


Figure 4-4 Simulation results for building with internal flows

5 Practical examples for analyzing airflow in projects

5.1 Strategies to increase the ventilation rate by incremental architectural modifications: An example

Here an example is taken to show how incremental modifications to the building design can have a big impact on the ventilation and airflow. The modifications do not represent fully featured architecturally developed designs, but they reveal technically how it is possible to significantly enhance the cross-ventilation air flow across a flat by small specific features on the façade.

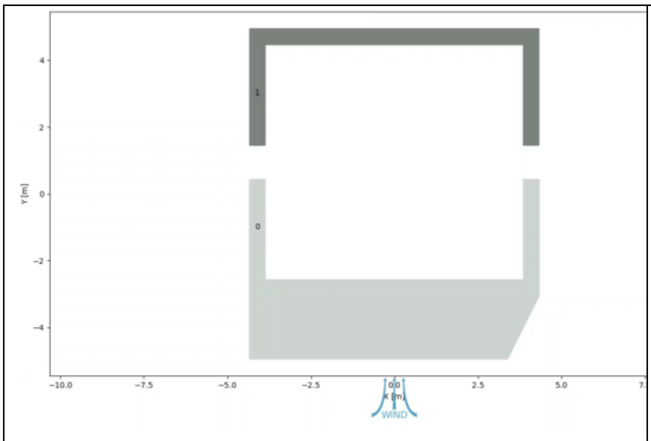
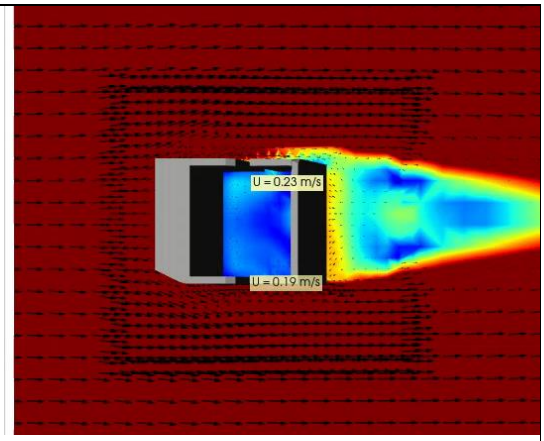
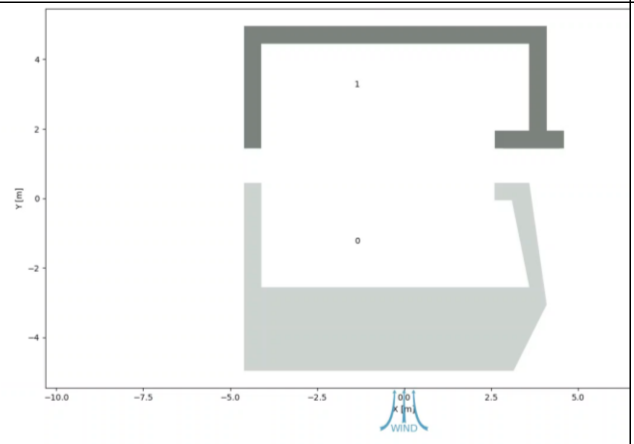
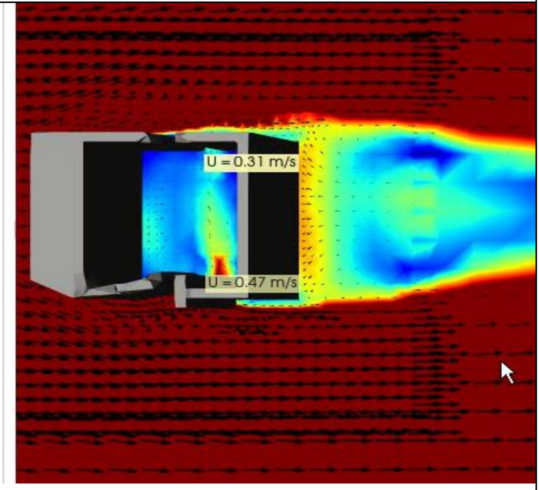
By working on the façade and building envelope, one can have a significant impact on the cross-ventilation achieved with the wind flow parallel to the building façades. This knowledge at the time of building design can be useful in improving natural wind driven ventilation.

5.1.1 Analyzing incremental benefits of façade elements by using Vayu Pravah

The analysis for the case is done through Vayu Pravah software and the results show that the software gives a good qualitative results and reasonable comparison between the different cases for the change in airflow with each increment.

5.1.2 Results

A comparison of the velocity magnitude (representative of the airflow in the building) shows the incremental changes and the consequent improved ventilation in the building. A slice of the result in each case is shown for comparison. (Building height : 10 m , slice at 5 m)

 Case 1 : Base case	 Resulting wind velocity in base case
 Case 2 : Adding a sloping element and Creation of a recess to have the wind entering it and gain pressure	 Simulation result with modified design

5.2 Wind parallel or perpendicular to the project façade

To expose the radical different ways to get air movement in a project, here is a case where all the buildings are perpendicular to the wind direction, and then the other extreme case where the buildings are “aligned” to the wind direction. As shown in the simulations, change in project design can change the air flow in the project to large extent.

These cases are being discussed for two layout cases here.

5.2.1 Case with buildings perpendicular to the wind direction

First is the example of a project where the wind falls perpendicular to the façade. As can be seen, although the buildings facing the wind direction receive good air flow, the rest of the project, including the buildings in the back receive poor ventilation.

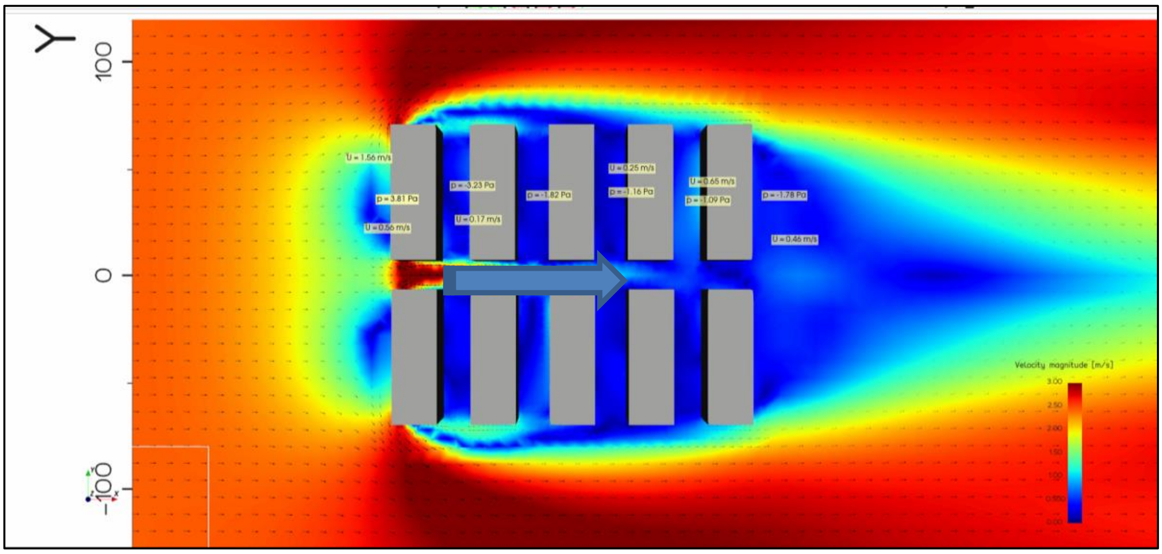


Figure 5-1: buildings massing perpendicular to the wind direction

In this case the first row of building is windward. The next rows are left with hardly any wind speed neither pressure difference from one façade to the other. The cross ventilation would not be efficient in this case for most of the buildings.

5.2.2 Case with the buildings massing parallel to the wind direction

This another extreme case with wind passing along the buildings. This example is not a real project, but it is described to explain the difference in the principle between the fully pressure driven natural ventilation (windward, leeward) and the case where the buildings long façades are more in line with the wind direction.

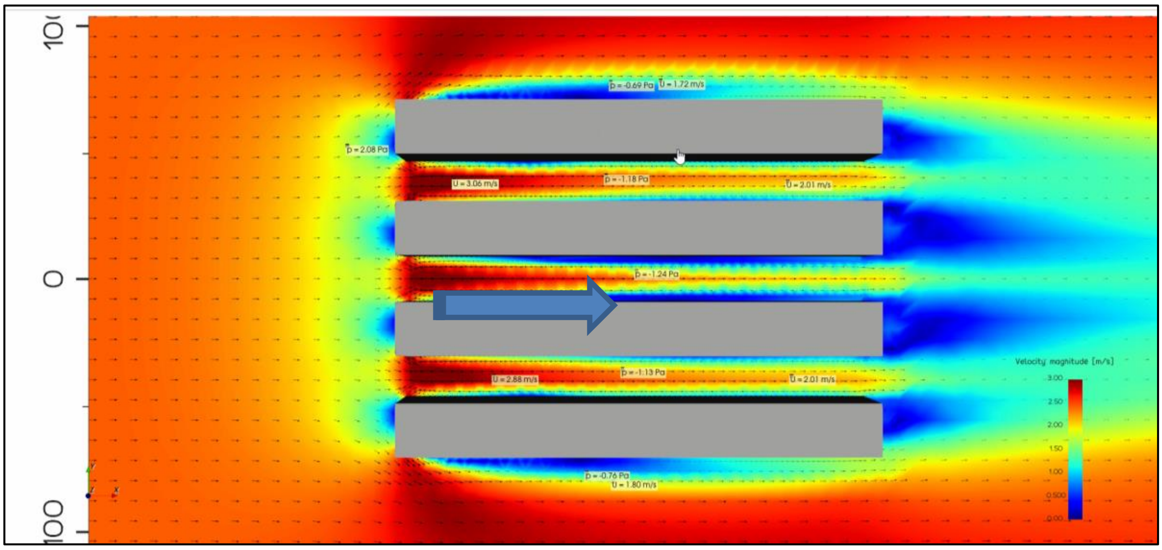


Figure 5-2: long facades aligned with the wind direction

In this case the usual way to consider the pressure difference between the façade as the driving force for the cross ventilation is no more relevant.

5.2.3 Mass building project with elements like openings with recess, inward inclinations, and fins

The next case is of a project with multiple buildings considering various openings in the buildings and the resulting airflow through the entire project. As discussed above, the orientation of the buildings vis-à-vis the direction of the wind can significantly change the resulting air flow and ventilation through the different buildings, especially the ones which are not directly facing the wind.

5.2.3.1 Project with buildings perpendicular to wind direction

This is the case where the buildings are perpendicular to the wind direction (90° N, 3m/s). The resulting wind velocity through the project can be seen in this simulation result.

Here it is seen that the buildings which are not directly facing the wind receive poor airflow and the natural cross ventilation is limited to the buildings directly facing the wind.

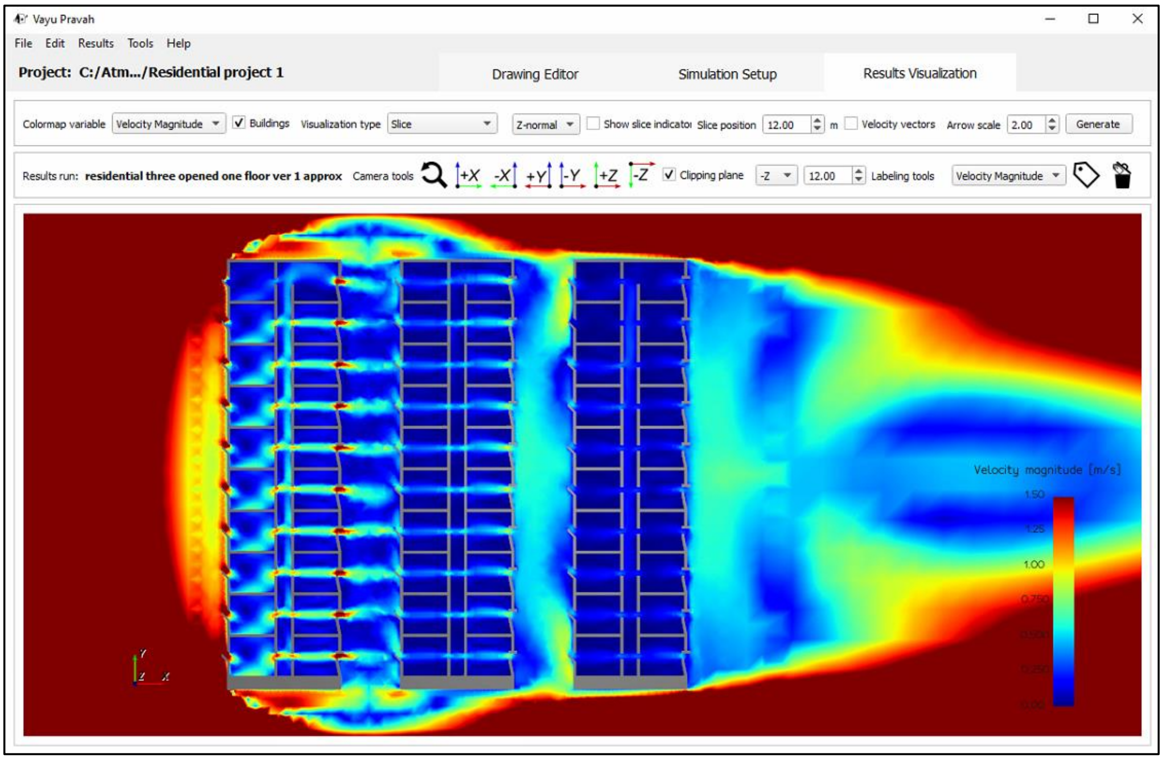


Figure 5-3 Project with wind perpendicular to facade, simulation for wind velocity magnitude, slice at 12 m height

5.2.3.2 Project with buildings almost parallel to wind direction

This is the case where the buildings are nearing the parallel direction to the wind direction.

When the project is subjected to wind parallel to its orientation (at 165` N, 3m/s) and a simulation is run, the results are as shown below. There is airflow throughout the project, even in the buildings which are not directly facing the wind. Hence there is good natural cross ventilation throughout the project.

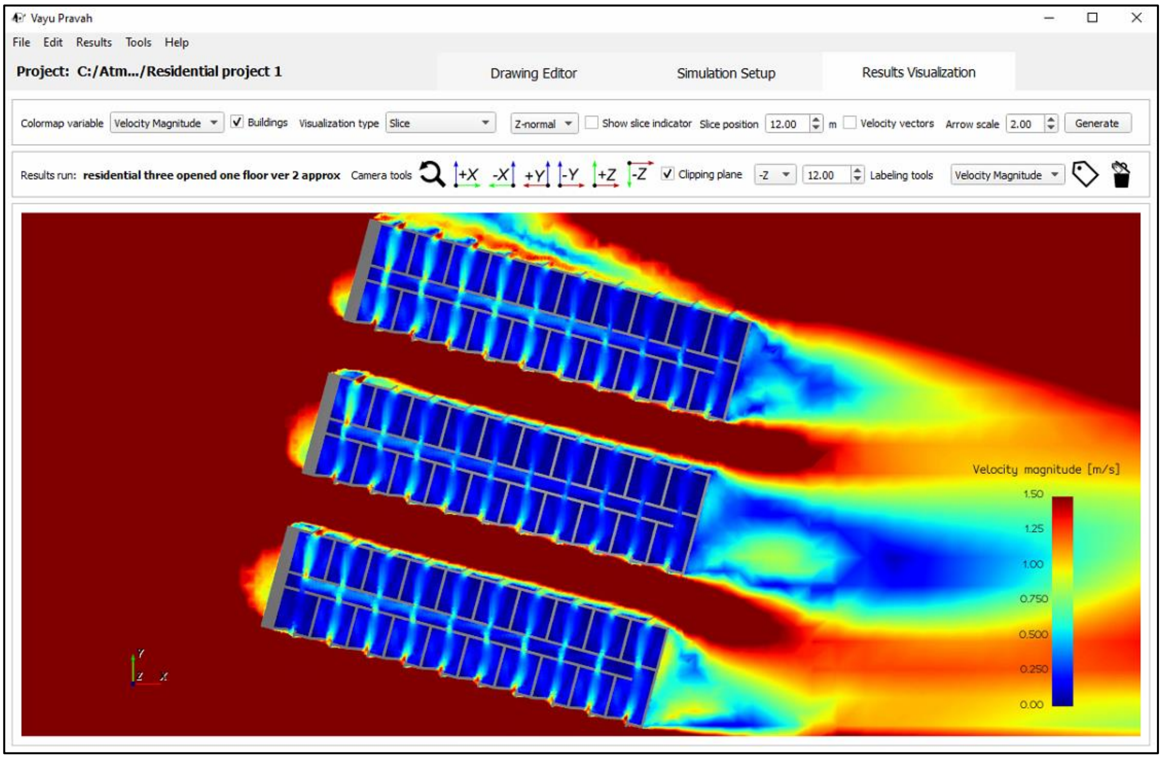


Figure 5-4 Buildings with buildings parallel to wind direction

Hence, we see with the help of the Vayu Pravah how the building orientation can affect the overall airflow in each of the buildings in a project.