



Disclaimer:

No portion (graphics or text) of this document may be reproduced, translated, or transmitted in any form or manner by any means – including but not limited to electronic copy, photocopy, or any other information storage and retrieval system – without explicit written consent from Bureau of Energy Efficiency, New Delhi

March, 2023

Contents

Contents.....	3
1 Introduction	5
1.1 Background Tool and installation	5
1.2 System requirements for installation	6
2 Functional user guide.....	7
2.1 Home Page and File Input.....	7
2.1.1 Project, case, and versions in Vayu Pravah.....	7
2.1.2 Open or Create a Project	8
2.1.3 Versions overview	9
2.1.4 Drawing Editor	11
2.2 Basic use of the Drawing Editor	15
2.2.1 Selecting a building	15
2.2.2 Select multiple buildings	16
2.2.3 Drawing Items	17
2.2.4 Selection.....	26
2.2.5 Tools section	27
2.2.6 Transform functions.....	36
2.3 Simulation Setup.....	37
2.3.1 Buildings.....	38
2.3.2 Prepare the simulation: Environment.....	41
2.3.3 During the simulation	43
2.4 Results Visualization	44
2.4.1 Select the Results Visualization pane.....	44
2.4.2 Selecting, preparing, and adjusting the View	46
2.4.3 Selecting the kind of results presentation	47
2.4.4 Visualization type.....	55
2.4.5 Clipping Plane.....	56
2.4.6 Taking Screenshot.....	56
2.4.7 Load results from another case in the same project	56
2.4.8 Visualizing the airflow through an opening	57
2.4.9 Generate pdfs reports.....	58
3 Example with buildings	61
4 Adding resistance.....	65
4.1 Simple equation for calculation of air across a window	65
4.1.1 Bernoulli Equation a refresher for an opened window without resistance	65

4.1.2	Example with resistance and fan	66
4.1.3	Example with a resistance like mosquito net	69

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 Tool and installation

To install Vayu Pravah, first install OpenFOAM using the supplied installer. Make sure to run it as administrator.

During the OpenFOAM installation, you can also install Microsoft MPI by clicking yes when prompted.

If you did not install MS MPI using the OpenFOAM installer, you can install it separately using the supplied installer. The installer should be run as administrator.

Vayu Pravah can now be installed. Run the installer as administrator.

When Vayu Pravah is launched for the first time, you will be asked to select the path where OpenFOAM and MS MPI have been installed. If they have been installed in their default location, the path will already be specified in the setup dialog, in which case you can simply press on “Next”. Otherwise, press on “browse” and select the path where you have installed OpenFOAM and MS MPI.

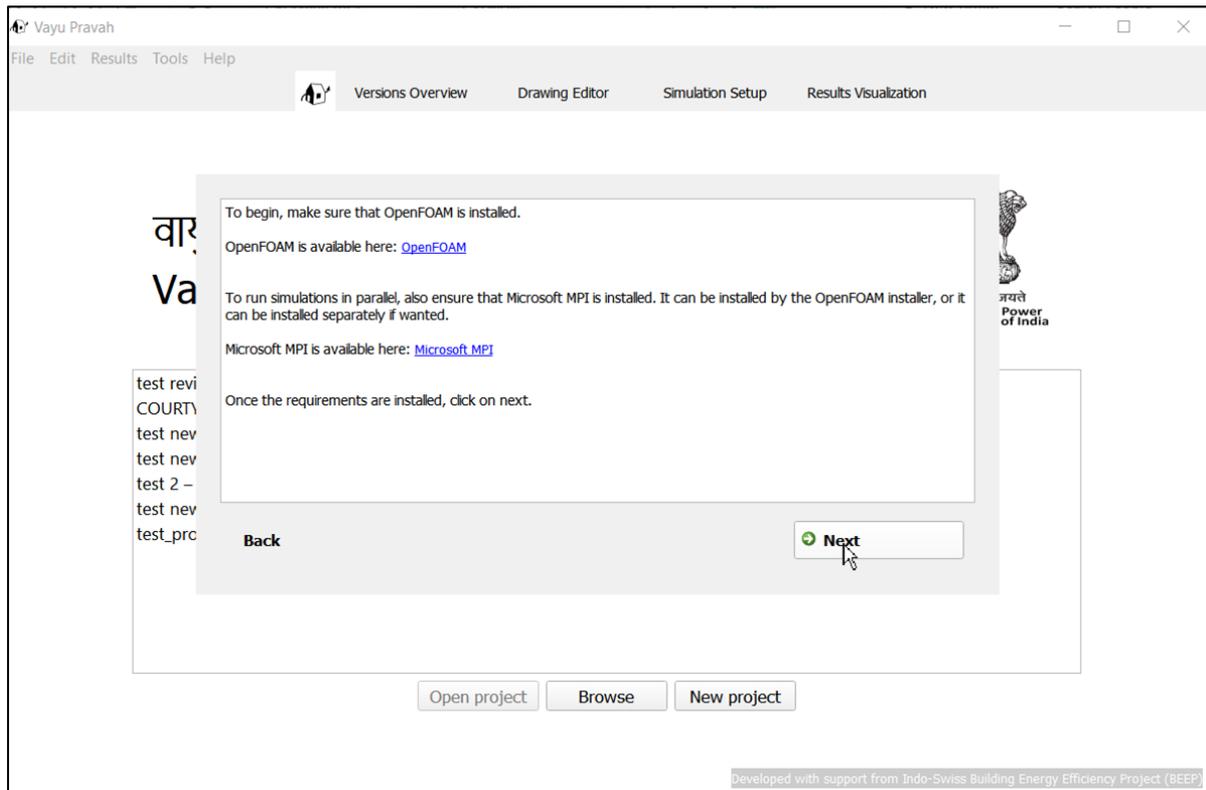


Figure 1-1 Running Vayu Pravah for the first time

Once the paths have been specified, you are ready to use Vayu Pravah. If for some reason you need to change the path to OpenFOAM and MS MPI, you can click on “Tools > Change OpenFOAM and MPI Path” from the top menu in Vayu Pravah.

1.2 System requirements for installation

The software can be installed on Windows 10 and Windows 11 systems.

To install the software, any previous version of Vayu Pravah in the system must be deleted before new installation.

Also, while operating Vayu Pravah, if other 3D modelling software are running in parallel in the system, they may interfere with the graphical interface of Vayu Pravah. In such cases, the user is advised to close these softwares and then work with Vayu Pravah.

2 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.

2.1 Home Page and File Input

At the home page, the user can start a new project or open an existing project. At any point during the project, the user can return to the home page by clicking on the Home icon .

2.1.1 Project, case, and versions in Vayu Pravah

In Vayu Pravah, 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 Versions Overview tab.

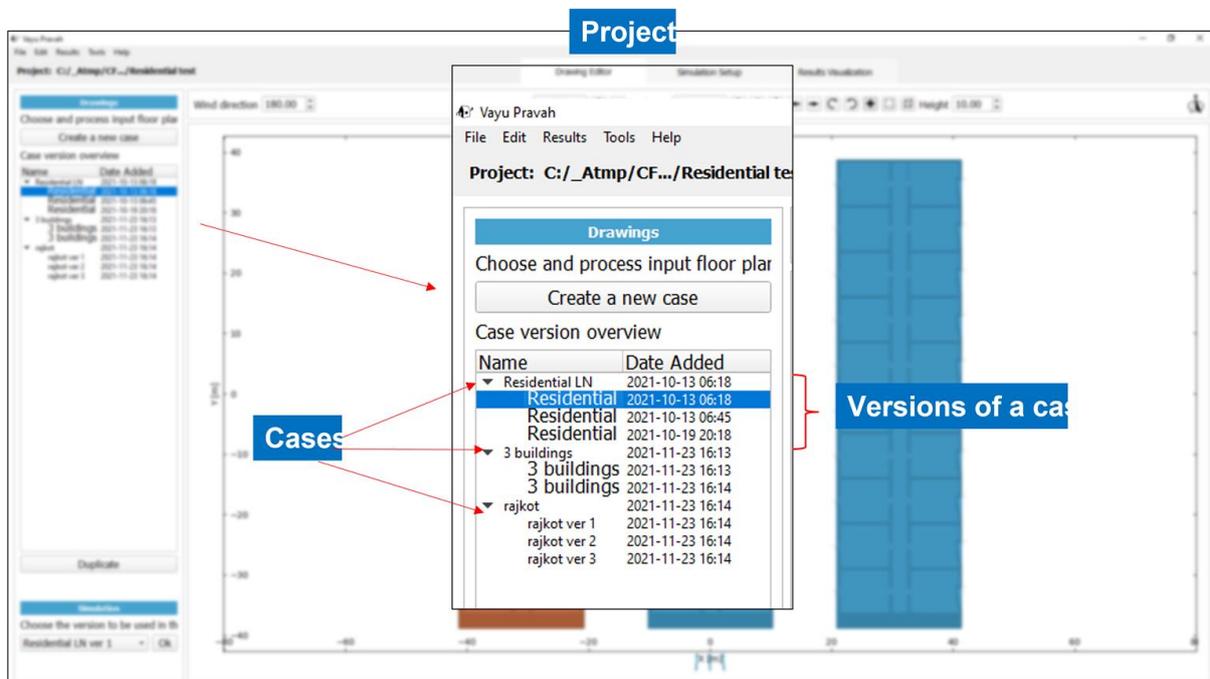


Figure 2-1 Project, case, and versions in Vayu Pravah. All the cases and their different versions are listed in the left-hand side panel in the Versions Overview tab

2.1.2 Open or Create a Project

A new project can be started by clicking on 'Create new project' tab. The user can also open 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 selecting the project name and clicking on 'Open Project'. The user can also specify another project to load by clicking on 'Browse' and selecting a project from the system.

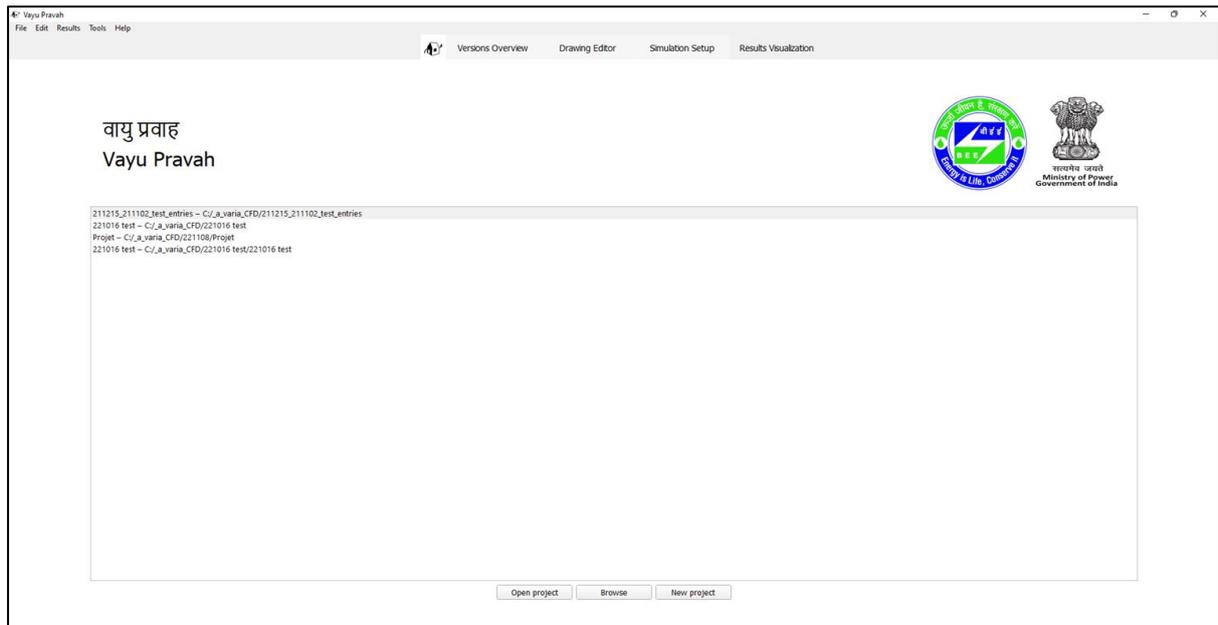


Figure 2-2 Home Page: create a new project or open an existing project

Once the project is created, it opens in the Versions Overview tab of the software.

2.1.3 Versions overview

The home page of the Vayu Pravah consists of four tabs namely Version Overview, the Drawing Editor, Simulation Setup and Results Visualisation. A project opens in the versions overview tab at the beginning.

All the different cases of a project are listed here, and the user can choose the case to work on from the left-hand side panel. The selected case then opens in the Drawing Editor. The user can also create new cases here, by clicking on the create new case, and saving it with a

name and a description (it is optional to add description). A case can also be deleted or duplicated here.

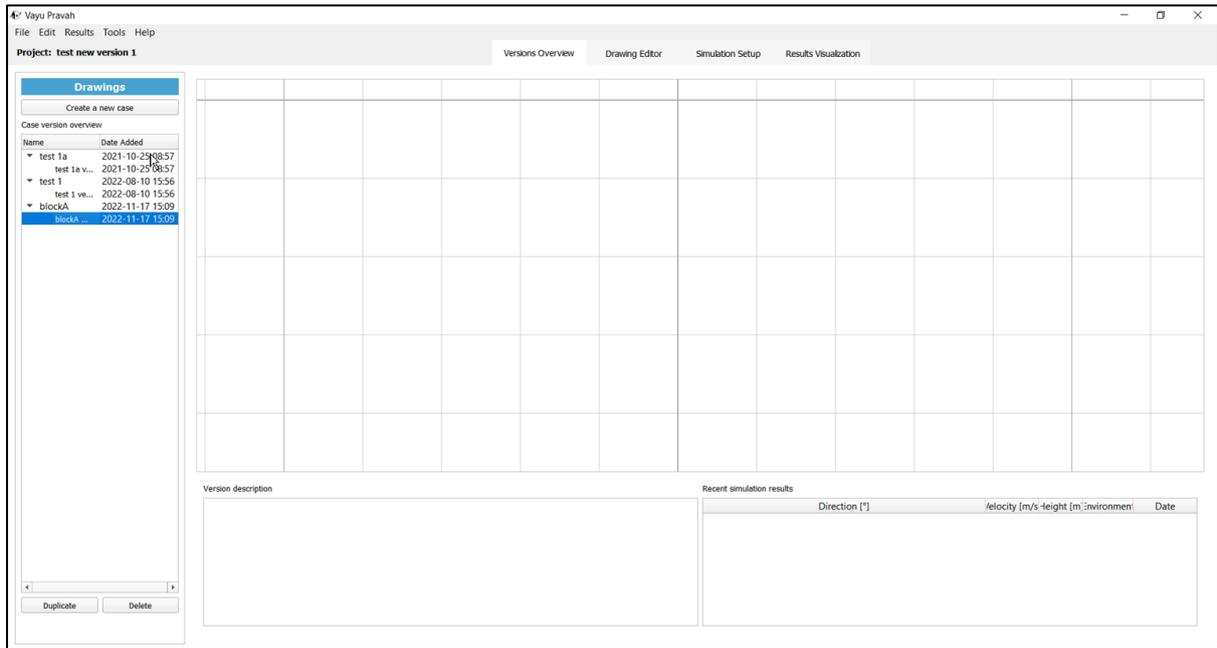


Figure 2-3 The Versions Overview tab

2.1.3.1 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 (optional), and then start working on it in the Drawing Editor.

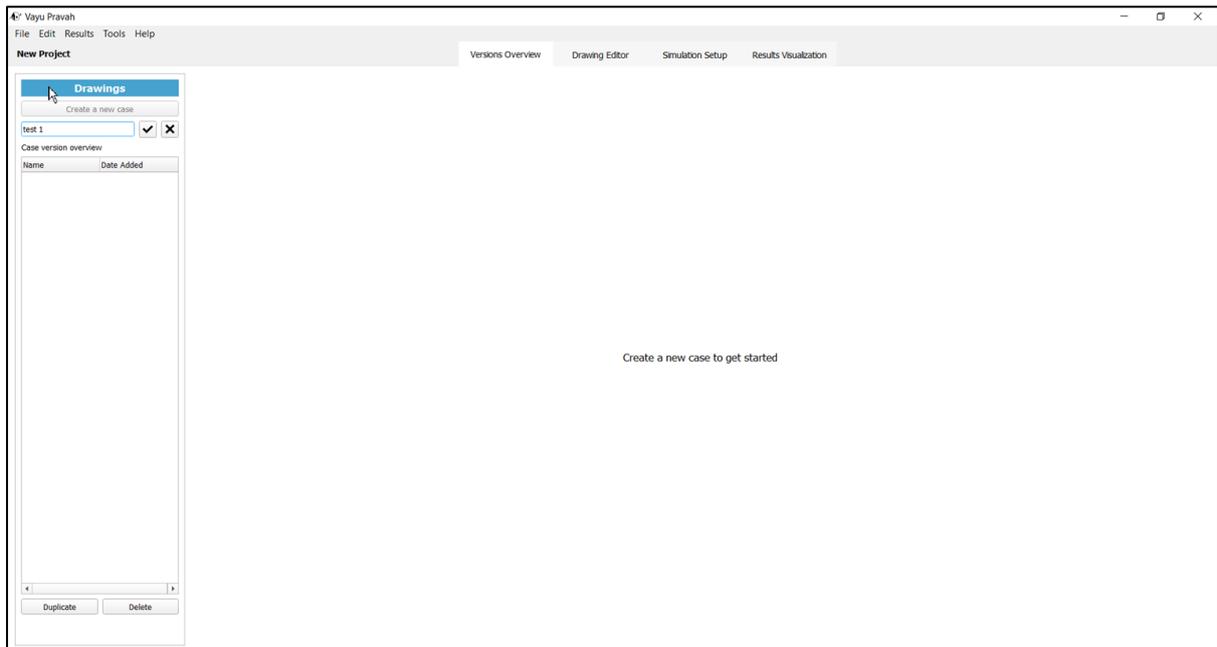


Figure 2-4 'Create a new case' prompts the user to name and save a new case

2.1.3.2 Duplicating and deleting 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 creates a copy of the selected version and creates a duplicate as a new version.

Clicking on Delete will delete the selected version.

1.1.1.1 Recent simulation results

The Versions Overview tab also has on the bottom right a list of the already prepared simulations from the current project. These are listed by the simulation parameters like wind direction, wind velocity, and the date of creation. The user can click on these to directly go to a pre-existing simulation in the project.

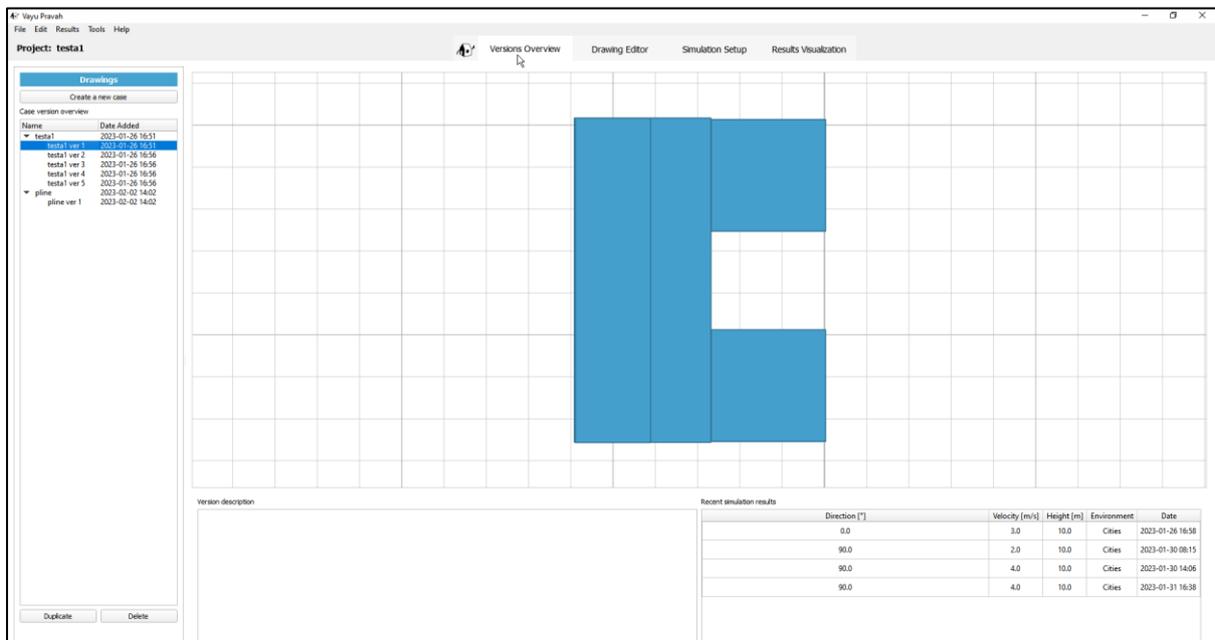


Figure 2-5 List of already done simulations of the selected version are shown in Recent simulation results on bottom right corner

2.1.4 Drawing Editor

Once a case has been created, the user can click on the Drawing editor to work on it.

2.1.4.1 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 clicking on the import button on the top menu. .

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.

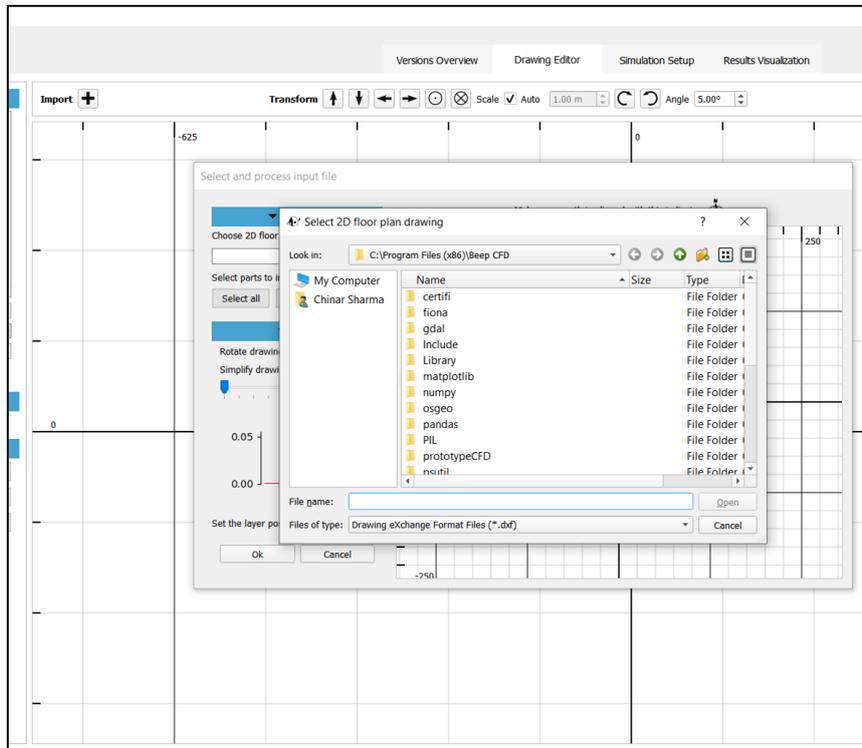


Figure 2-6 Importing a dxf file in a case

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 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).

2.1.4.2 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).

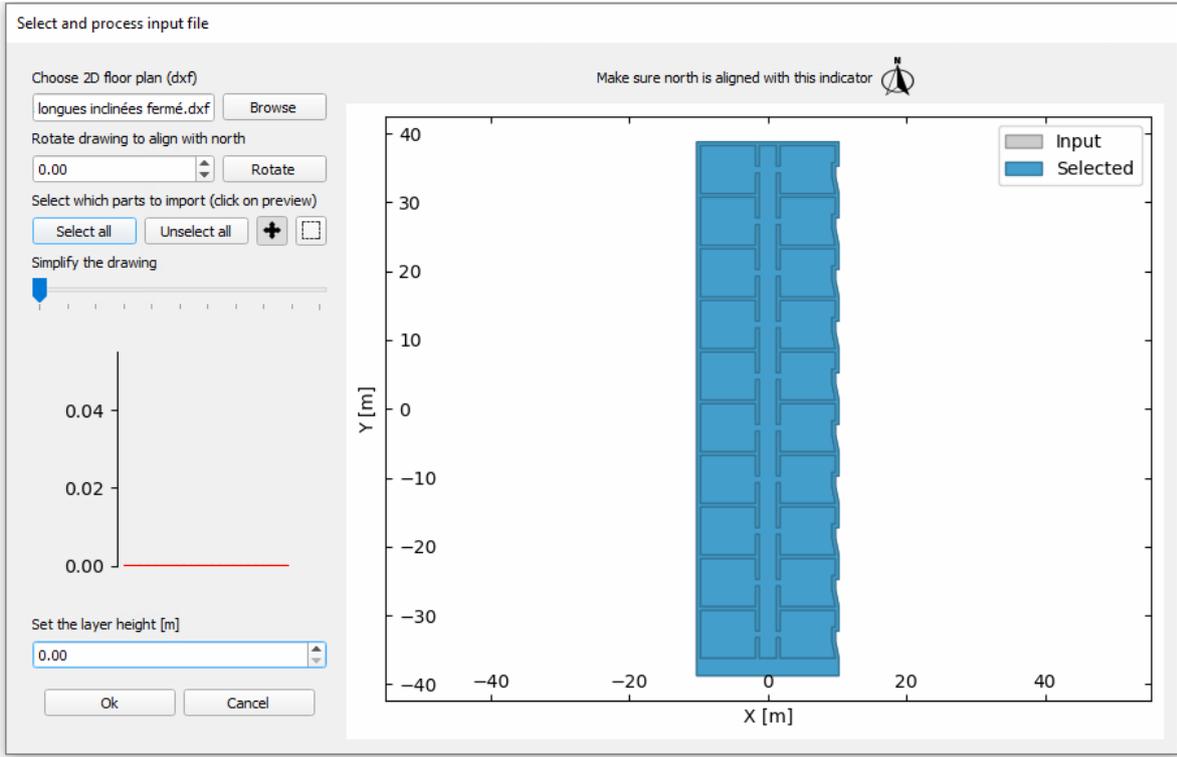


Figure 2-7 Importing the entire building (in blue)

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

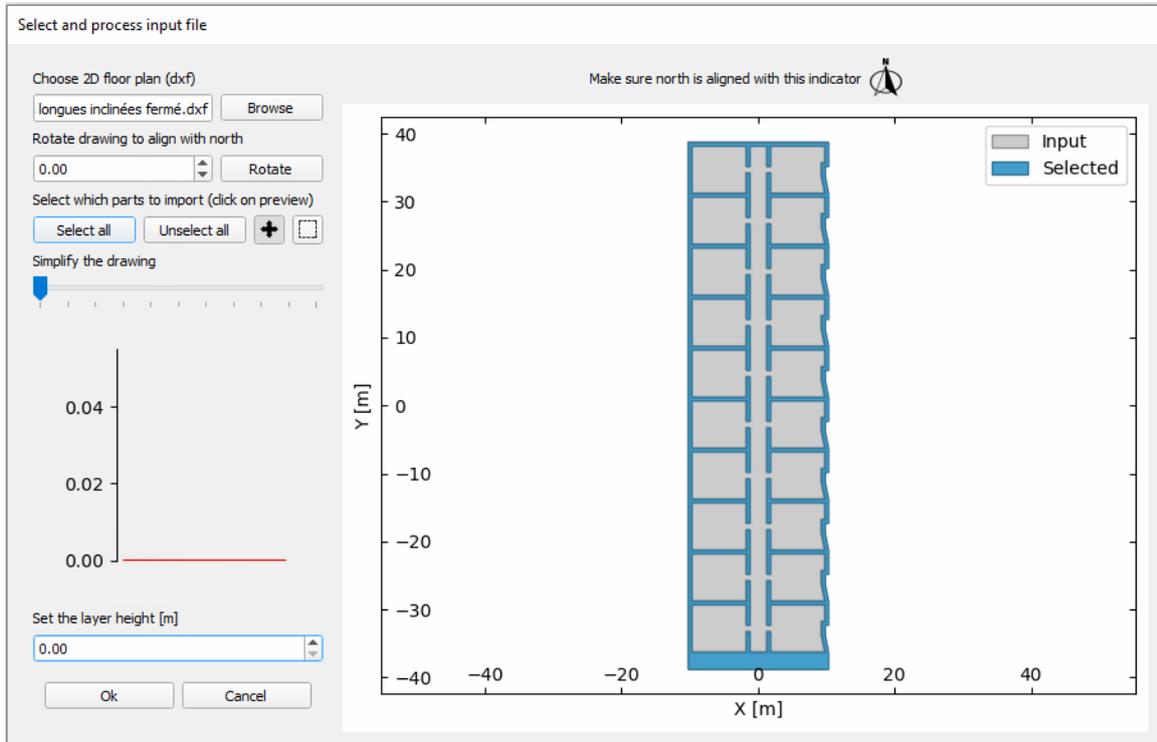


Figure 2-8 Selecting only a part of the dxf

2.1.4.3 Building selection

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

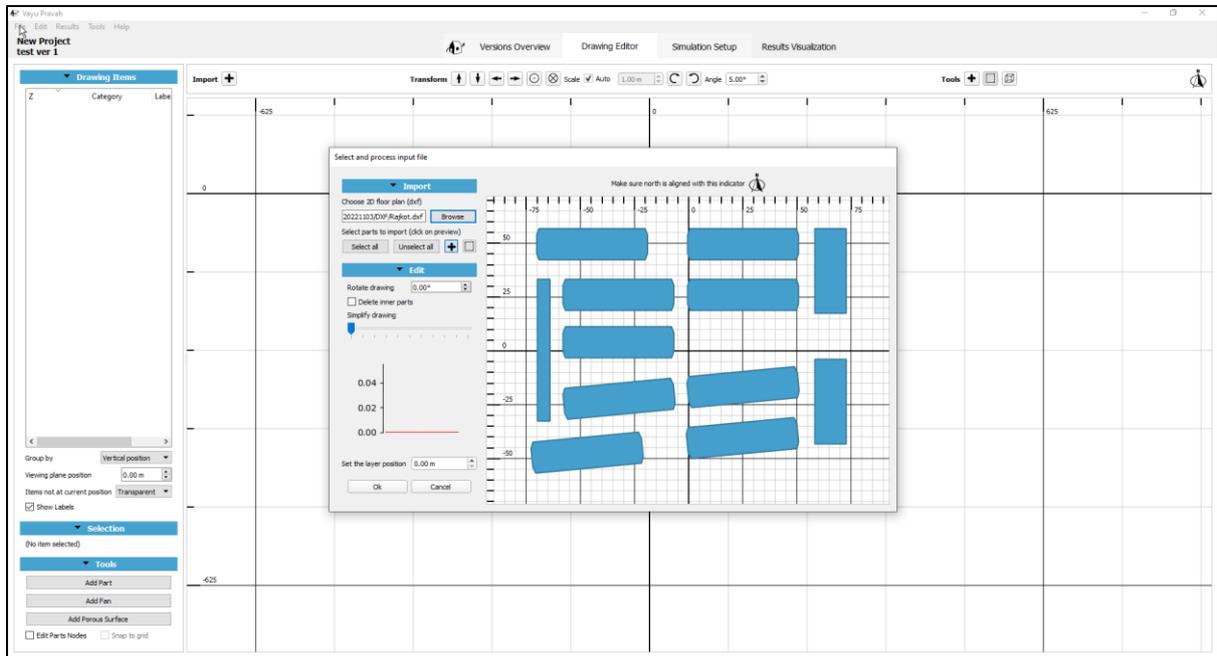


Figure 2-9 Loading a new file: By default, all the buildings have been selected (in blue)

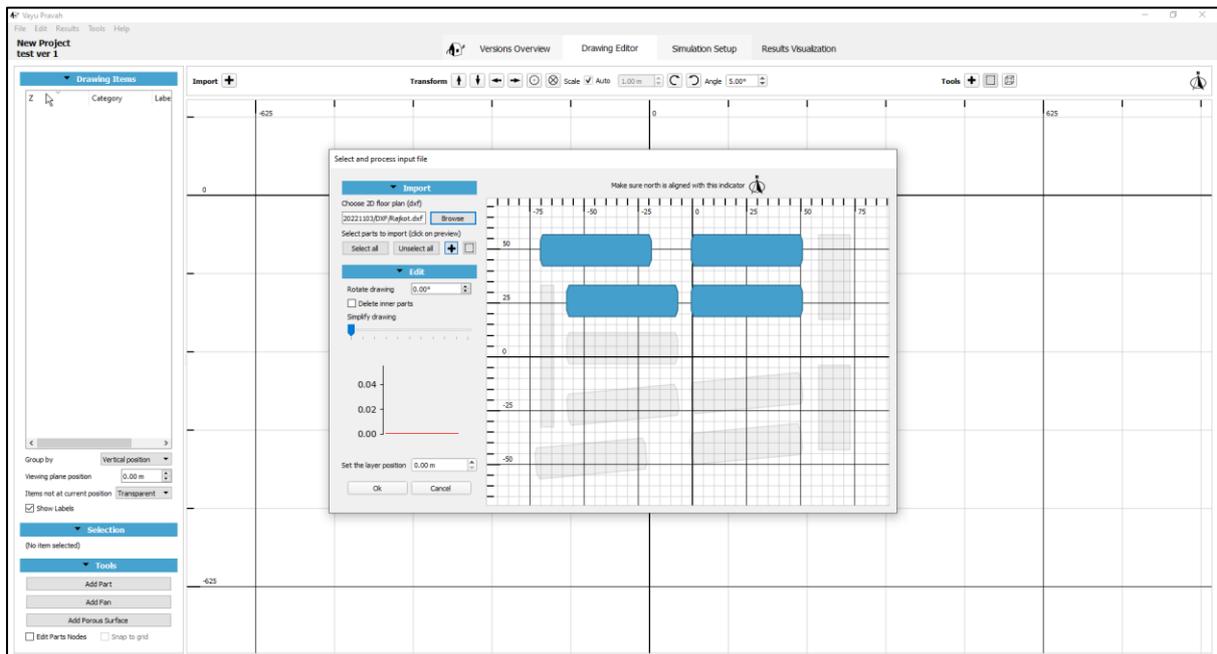


Figure 2-10 Manually selecting part of the dxf to import in the plan

The drawing can also be rotated by a desired angle at this point.

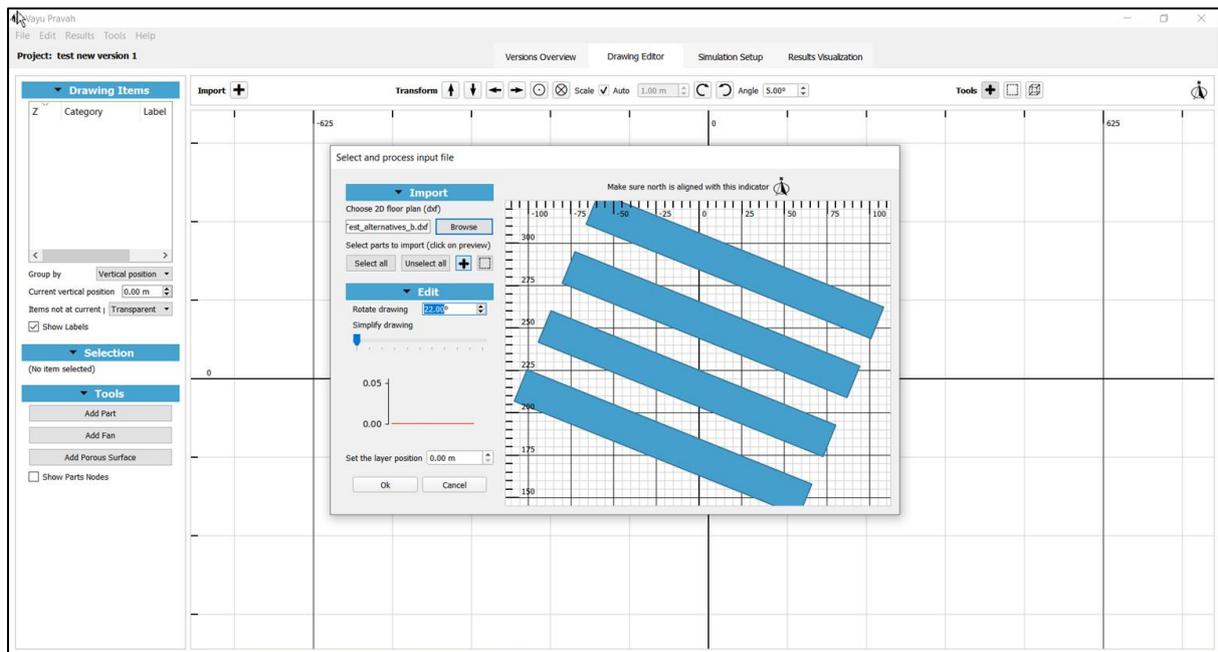


Figure 2-11 Rotating the dxf by specifying the angle of rotation

Press OK to load the drawing in the Drawing Editor.

2.2 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.

The North is indicated by the pointer on the top right hand side icon  in the Drawing Editor

2.2.1 Selecting a building

In a particular drawing, the user can select a building by clicking on it or by clicking on its part number in the left-hand side panel. The selected building in the dxf file is highlighted in brown in contrast to the unselected buildings in blue. It is on this building that all the changes are done.

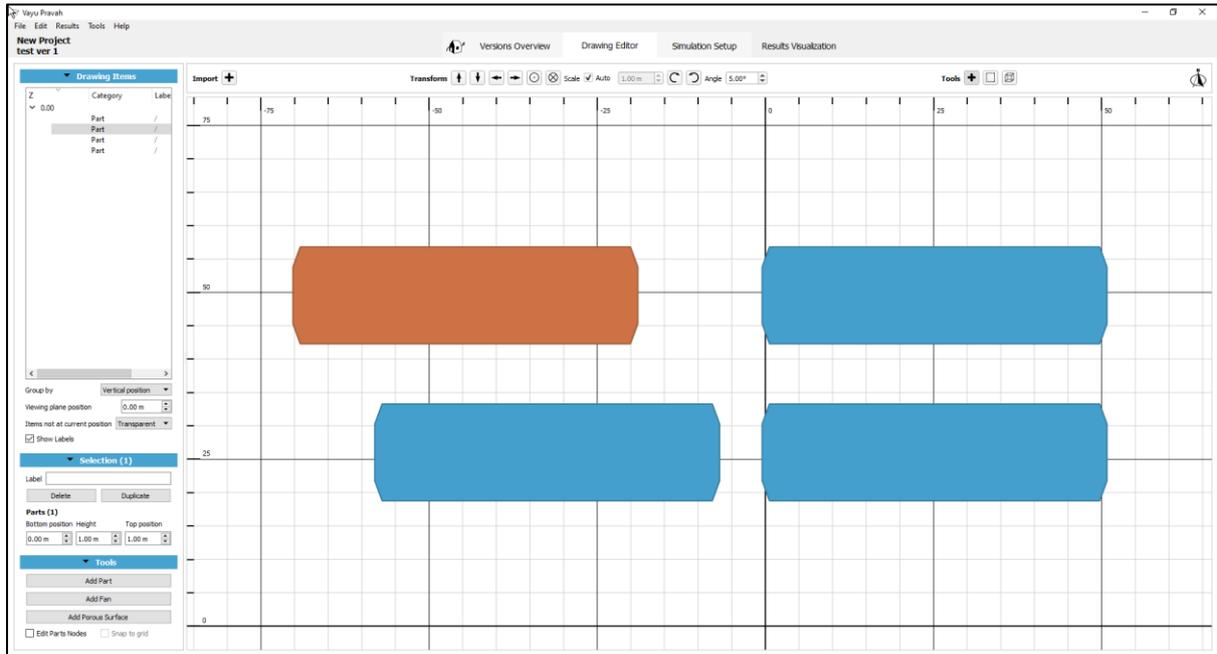


Figure 2-12 Selected building is highlighted

2.2.2 Select multiple buildings

By clicking on different buildings along with the Control key, multiple buildings can be selected. Multiple buildings can also be selected in the transformation toolbar, as explained later.

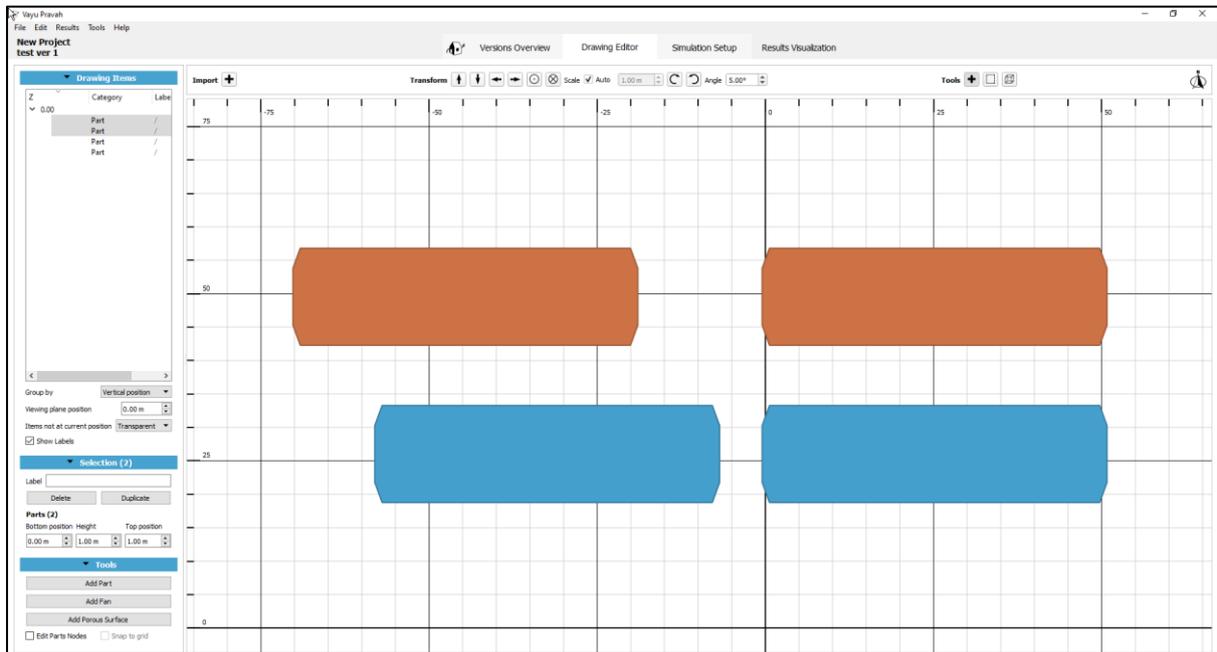


Figure 2-13 Selecting multiple buildings

2.2.3 Drawing Items

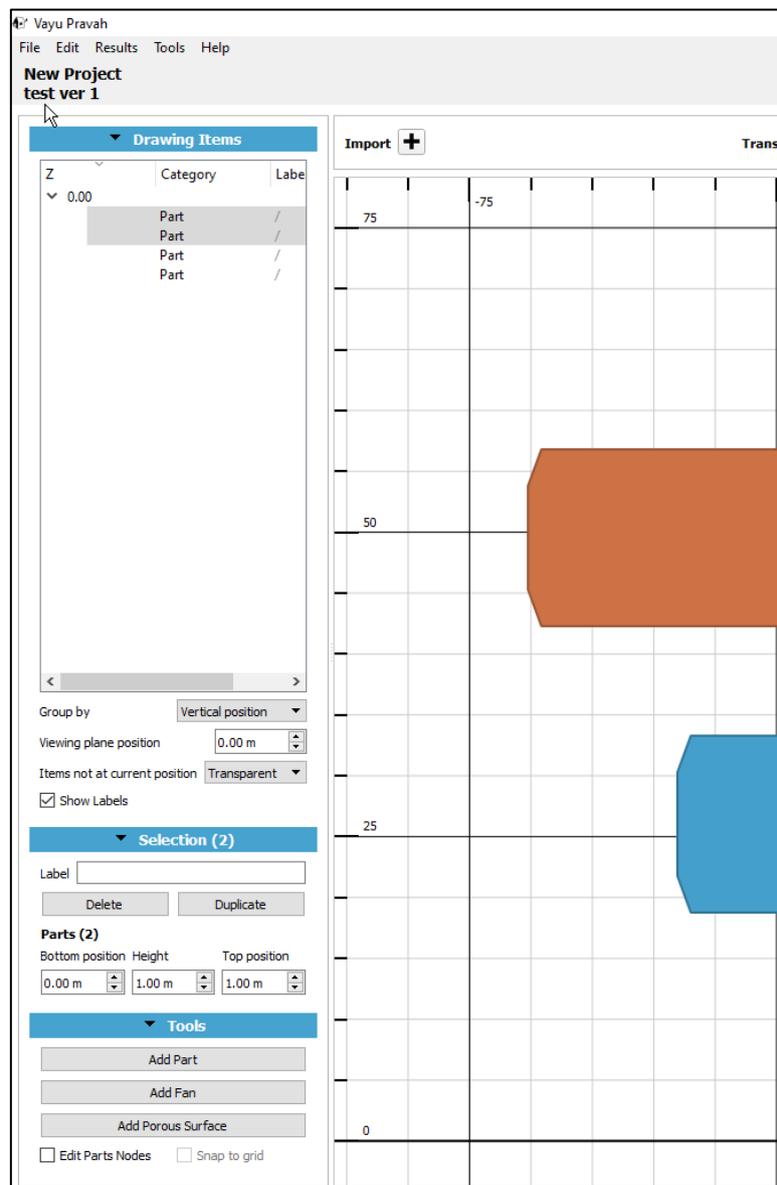
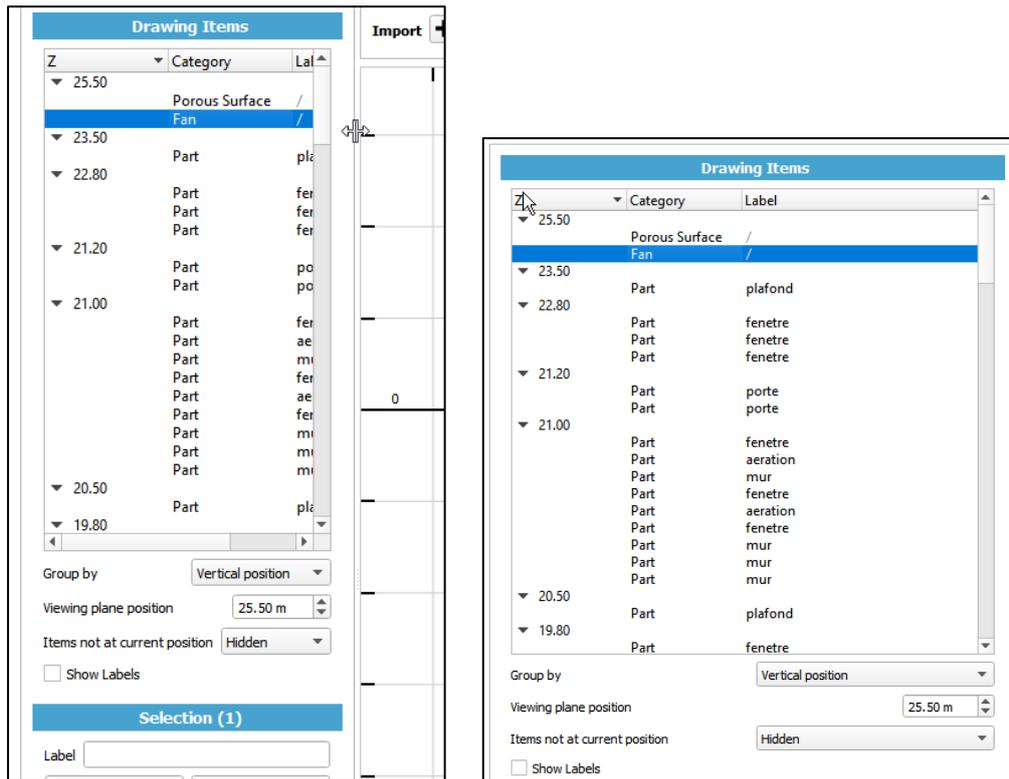


Figure 2-14 Drawing Items menu on the left-hand side of the Drawing Editor

On the topmost of the left-hand side panel is the drawing items menu. In this list, all the parts of the different *dxf* imported in the case are listed. Each closed polygon is treated as a separate part. The parts, by default, are grouped based on their vertical position on the plane.

The user can adjust the width of the left-hand side panel by clicking on the edge and dragging the cursor to desired width.



Choose the result to load

Version	Direction (°)	Velocity (m/s)	Height (m)	Environment	Date
Batiment Triple ver 1	170.0	1.5	10.0	Cities	2022-09-09 15:49
Batiment Triple ver 1	-10.0	1.0	10.0	Cities	2022-10-07 16:05
Rajkot ver 1	270.0	3.01	10.0	Cities	2023-02-02 15:17
Rajkot ver 1	270.0	0.01	10.0	Cities	2022-11-01 18:03
Rajkot ver 2	270.0	0.01	10.0	Cities	2022-11-07 15:36
Rajkot ver 4	270.0	0.01	10.0	Cities	2022-11-10 15:19
Rajkot ver 5	270.0	0.01	10.0	Cities	2022-12-09 15:37
Shaft Test ver 6	0.0	0.1	10.0	Cities	2022-12-18 01:52
Test ver 7	90.0	2.0	3.0	Cities	2023-01-10 15:46
Test ver 9	90.0	2.0	3.0	Cities	2023-01-10 15:52

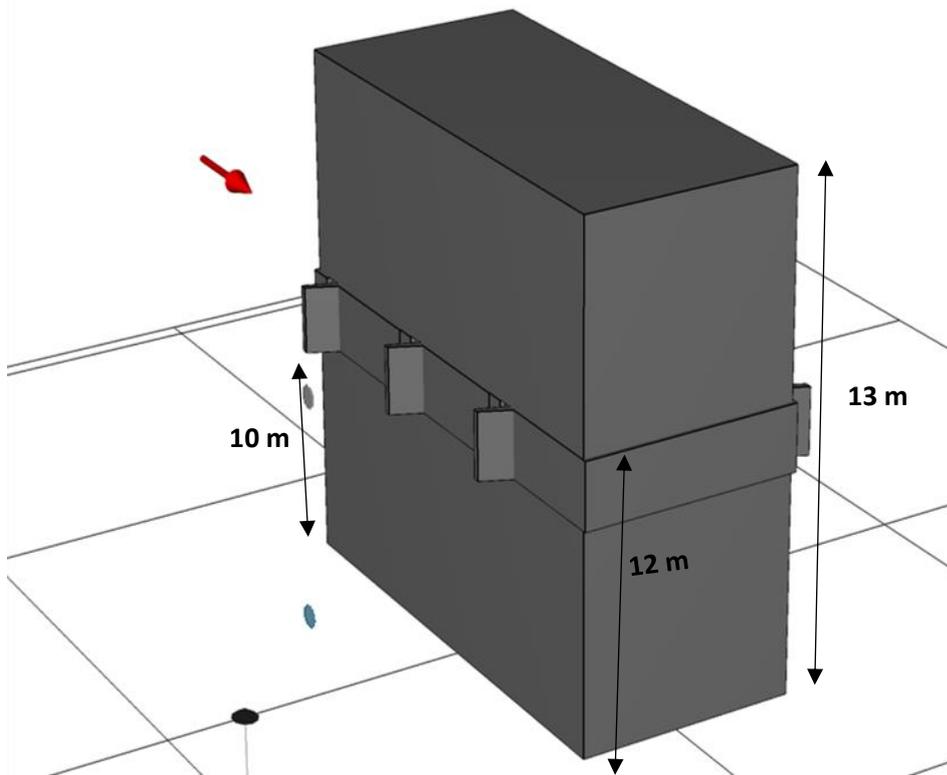
Figure 2-15 Adjusting the width of columns in different windows like Case versions overview, Drawing items, and results window

There is the labels column which mentions any label that has been given to a particular part. The vertical position and label are explained in detail below.

2.2.3.1 Group by

By default, the parts are grouped by planes, i.e. based on their bottom vertical positions, so that drawings on a plane are grouped together. However, the user can also group them basis their label or their category by selecting it from the drop-down menu.

For example, consider the building as shown below.



To create this building in the drawing editor, the user can add multiple windows, walls, a base, and a roof. These different parts will be added at different vertical positions from the ground (concept of vertical position is explained in detail later).

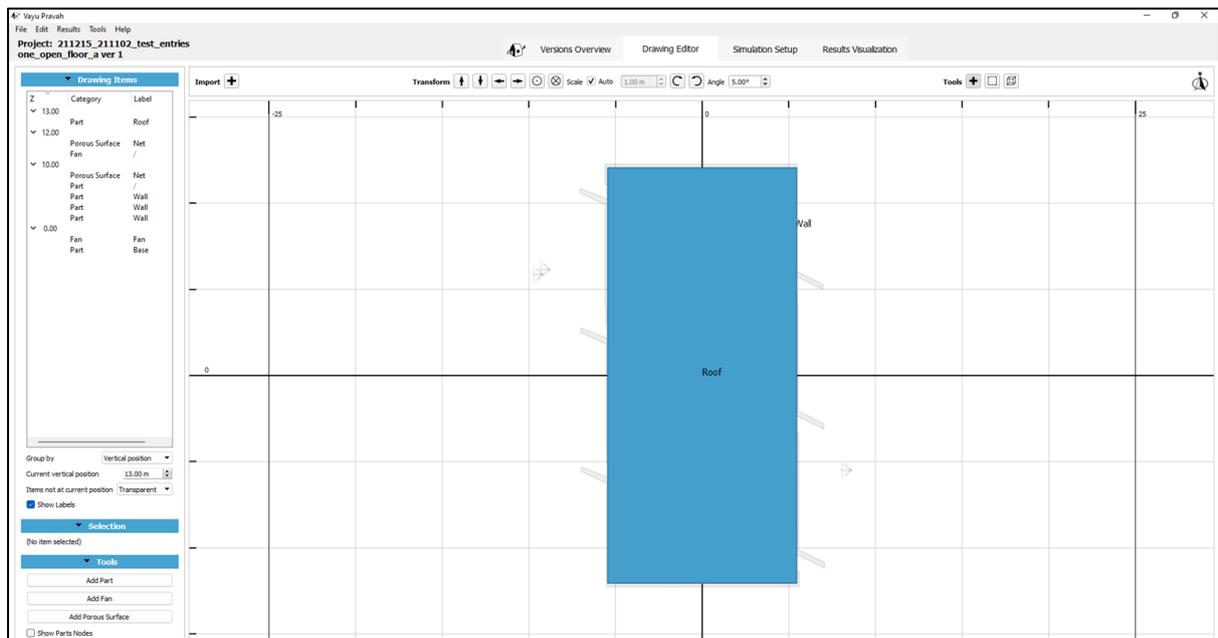


Figure 2-15 Building a) as seen in the Drawing Editor

Here, the different parts of the building are grouped by their vertical position from the reference. So, some parts are 0m from the reference (at the base or the bottom), some at 10

m, and so on. Again, some parts are labelled as wall, roof, fan, or porous surface (more on Labels in next sections). Hence, the user can choose to segregate the items by their vertical position from reference, their label, or their category.

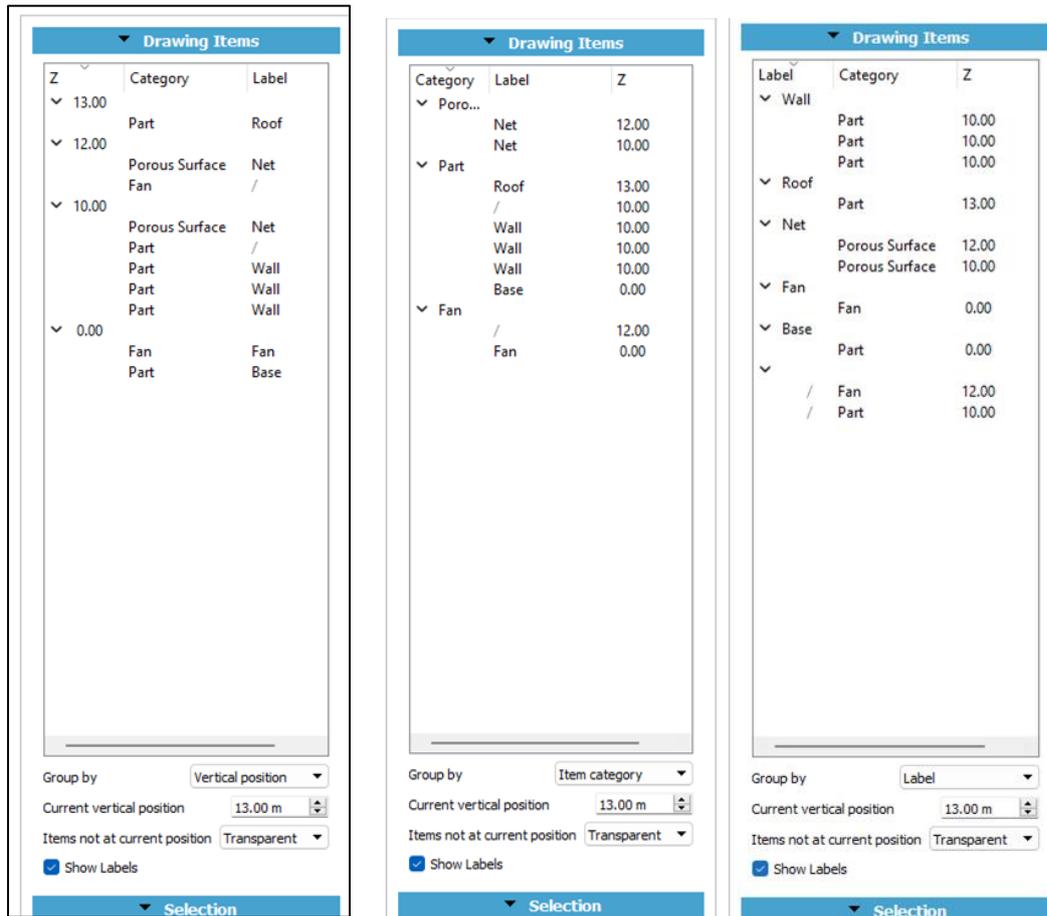


Figure 2-16 Grouping the parts by vertical position, or category or label

2.2.3.2 Viewing plane position

This specifies the current vertical position which is being viewed in the Drawing Editor

Vayu Pravah allows the user to specify for each part, its bottom position, and its height.

Bottom position:

The bottom position determines the vertical distance of the base of the part, from the reference 0 of the drawing editor.

For example, in case of block A here, the bottom position is 0m as it is on the reference 0m of the drawing editor. Its height is 6 m as shown in the figure. In case of block B, the bottom position is 8m and the height is 4m.

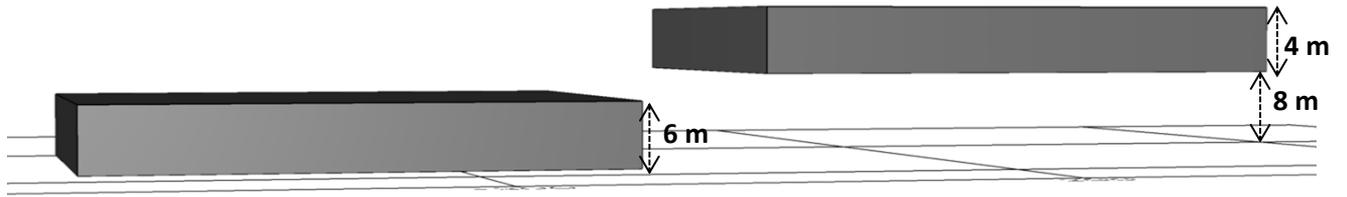


Figure 2-17 Two blocks with different bottom positions (from reference 0m) and different heights

For these blocks, the bottom position and the height should be defined are as follows

	Bottom position	Height	Top position (Automatically calculated as bottom position + height)
Block A	0 m	6m	6m
Block B	8m	4m	12m

As another example, consider a design as shown in fig below.

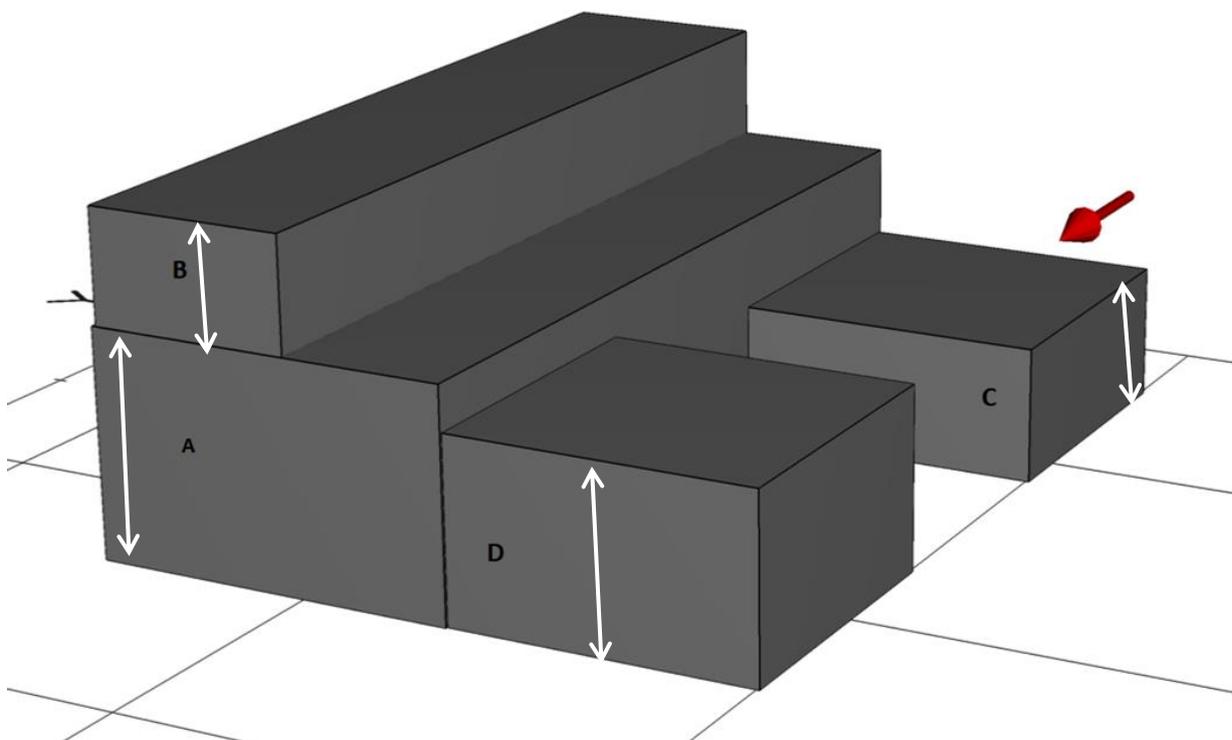


Figure 2-18 Building with multiple blocks at different bottom positions and different heights

The design is made with four blocks, A, B, C, D. Block A, C and D are at the base, and have different heights. Block B is placed on block A. To make this building, blocks A, B and C are entered at bottom position 0 m (reference base), and their heights are entered. Block B is added on top of Block A and hence its bottom position is not 0 m but 10 m (height of Block A).

To design this building in the drawing editor, the parameters of each block are added as shown in the figures below.

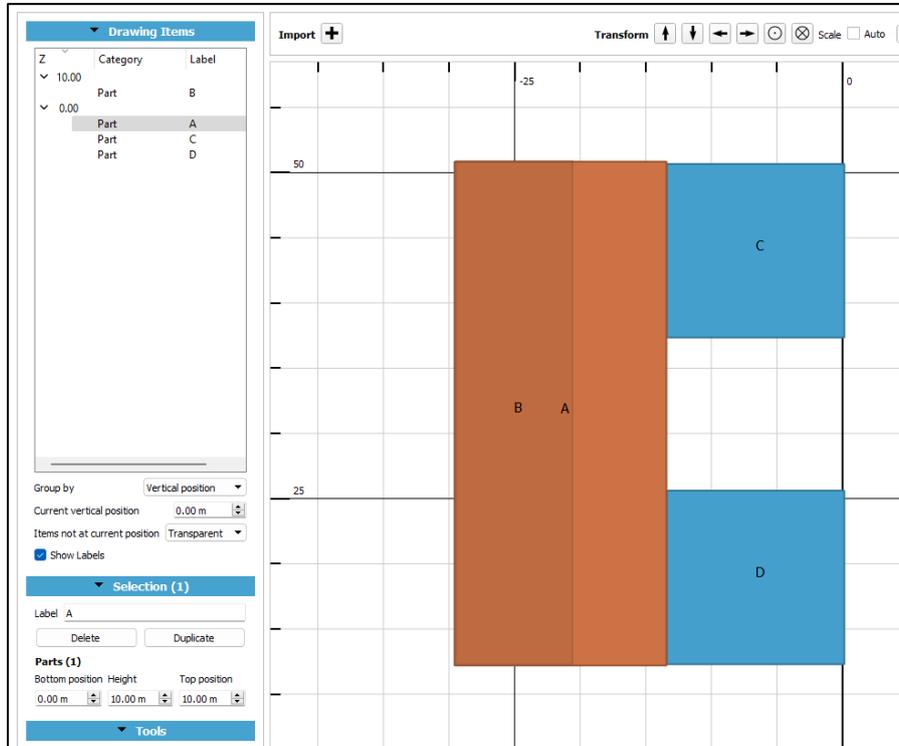


Figure 2-19 Block A, bottom position 0 m, height 10 m

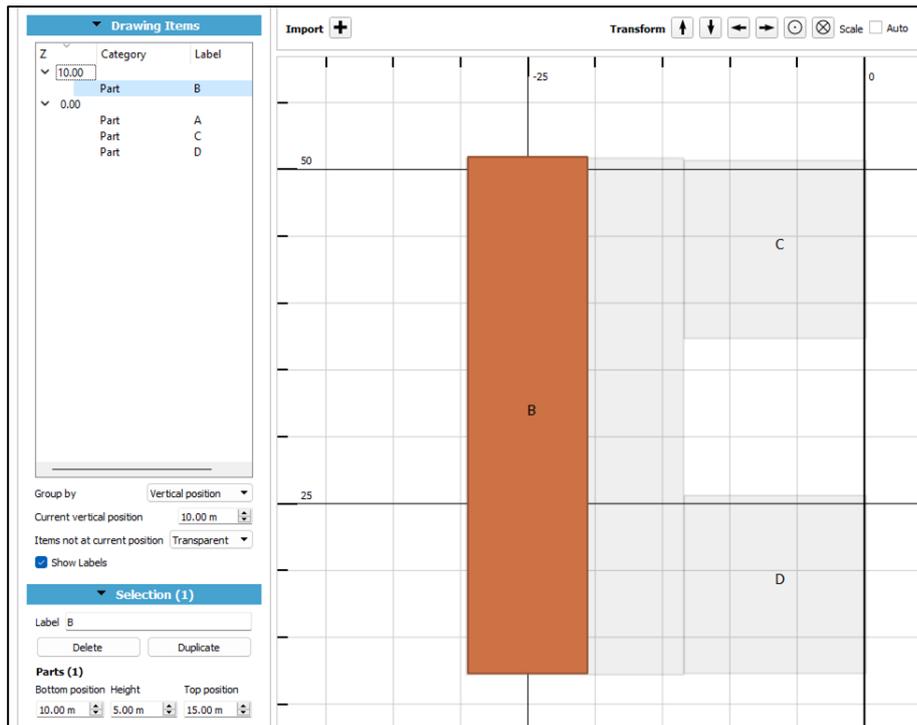


Figure 2-20 Block B, bottom position 10m, height 5 m

1.1.1.2 Viewing plane position

This gives the vertical position of the current plane being viewed in the drawing editor. The user can add parts at different vertical positions, creating different planes and this field shows the height (from ground) currently being seen.

2.2.3.3 Items not at current plane

Here, the user can choose to either hide, make transparent or opaque the part of the building, which is not at the current plane, by choosing from the drop-down menu. Take the example of the figure above again. Here, there are parts of the drawing at two planes, a) at 0m and the other at 10m .

While viewing the parts at 0m, the user can choose how to see the parts of the other plane.

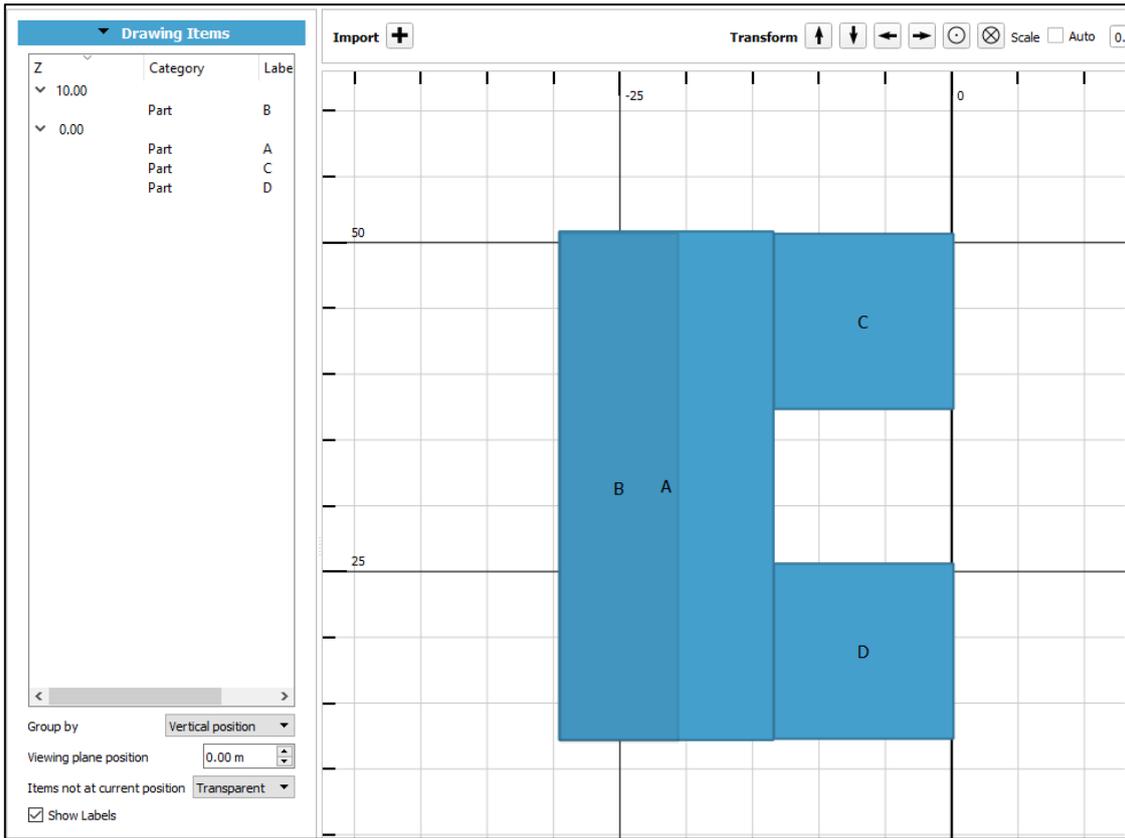


Figure 2-21 Items not on the same plane (in this case, block B) are transparent

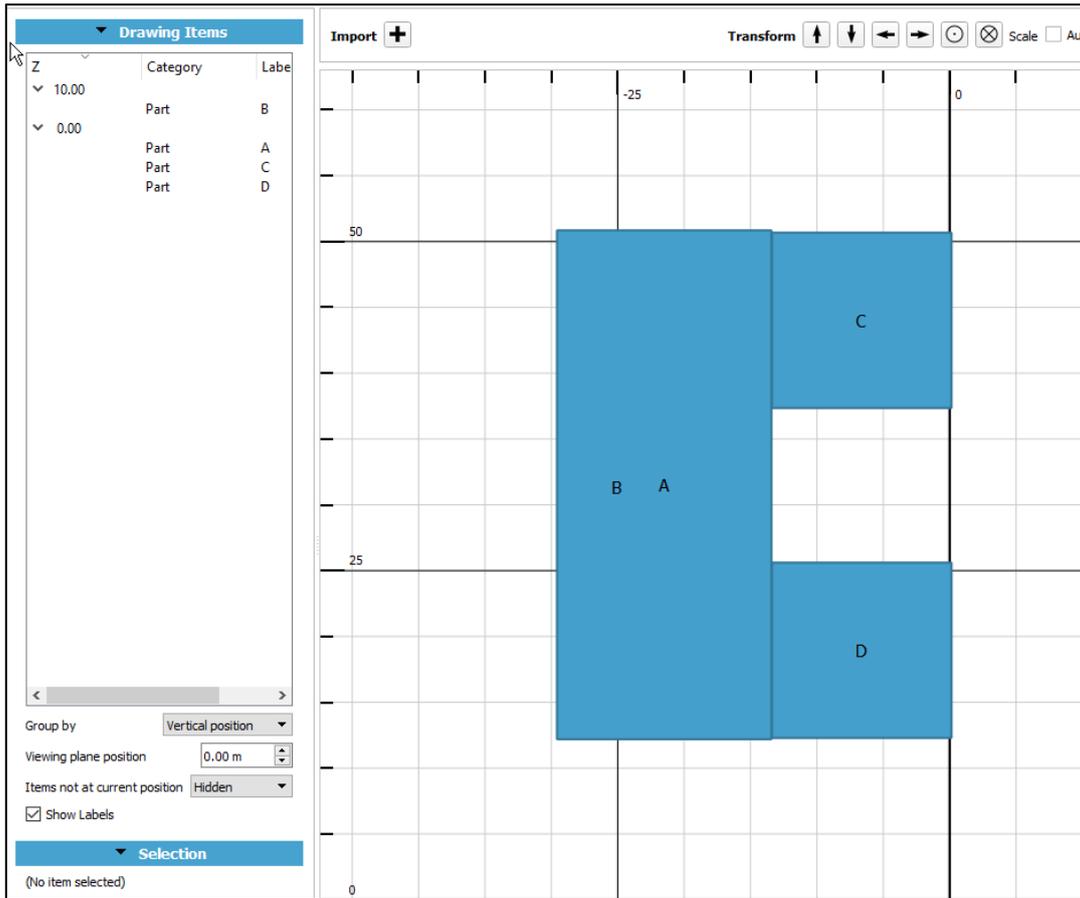


Figure 2-22 Items not on the same plane are hidden

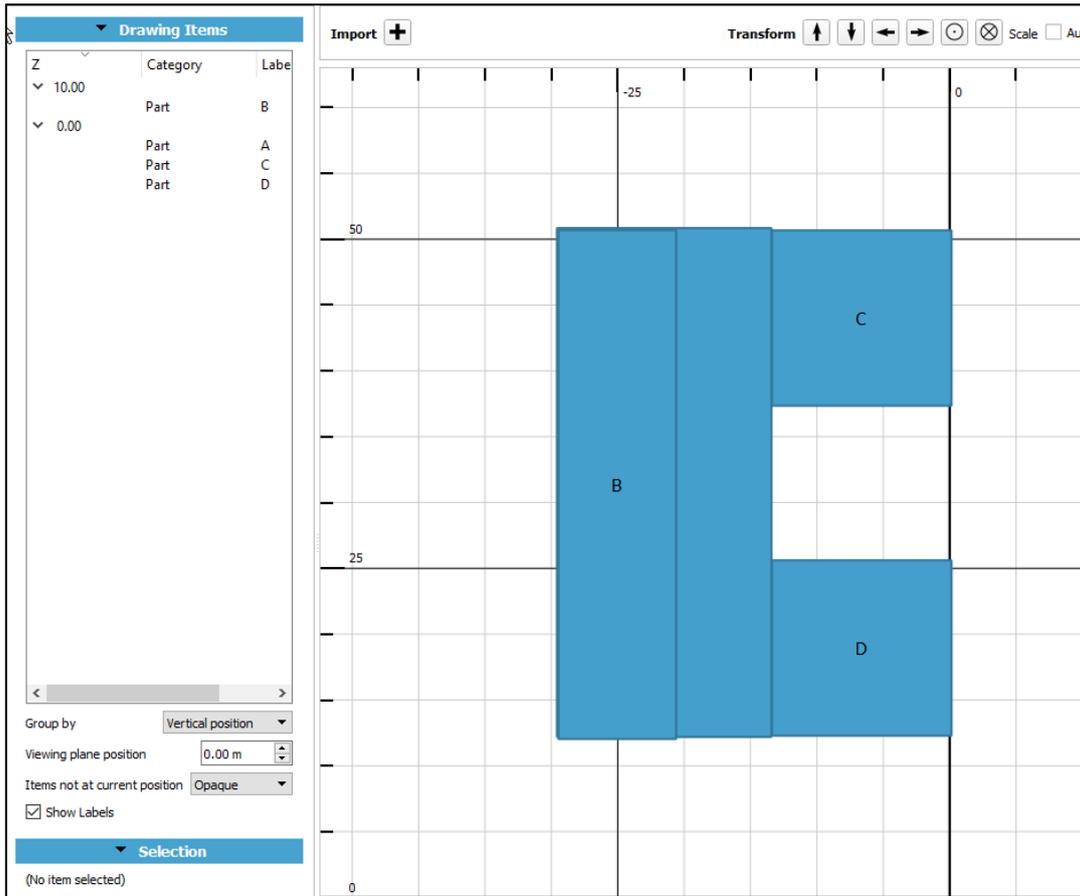


Figure 2-23 items not on the same plane are opaque

2.2.4 Selection

The selection section shows the current part selected in the drawing. The number in the bracket next to the heading Selection (n) shows the number of parts currently selected. Here the user can define some properties of the selected parts.

2.2.4.1 Adding label

In the label column, the user can add a relevant label to the selected part(s) for better identification and sorting of parts. The label can be entered in the field Label. Once a part is labelled, the label appears in the drawing.

2.2.4.2 Show/Hide Label

In the drawing editor, the user can choose to see or not these labels, by checking the box 'Show Labels'. If not required to be shown in the drawing, the user can uncheck the Label box in the Selection section. This will hide the label in the drawing.

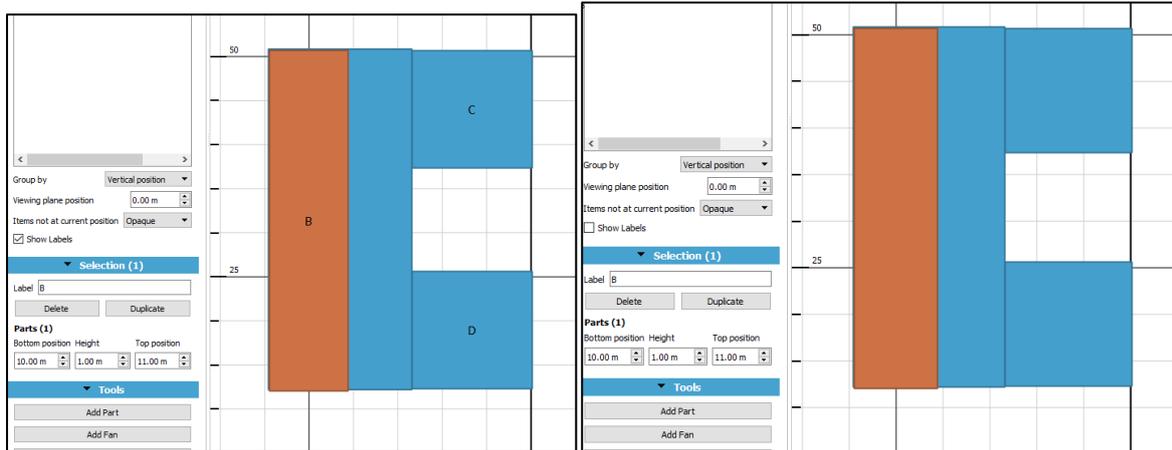


Figure 2-24 The label of part B is shown or hidden depending upon the check box Show Label

2.2.4.3 Delete and duplicate

The delete button is used to delete the selected part.

The duplicate button creates a duplicate of the selected part. The duplicate part is superimposed on the selected part and can be moved around by selecting it and using the arrow buttons in the top panel.

2.2.4.4 Bottom position, top position, and height

Here the user can specify the height of the selected part(s) and its bottom position (reference position). The top position is automatically calculated based on the height and the bottom position. By default, the bottom position is at 0m.

Thus, the user can create different planes by adding parts at different heights from the bottom and can look at these planes separately.

When parts of one plane are being worked on in the editor, the user can choose to either hide, or make transparent or opaque the other levels of the building.

2.2.5 Tools section

In this new version of Vayu Pravah, the user can add a new part, a fan, or a porous surface (like a mosquito net) to the building and analyse their effect on the air movement in the building.

2.2.5.1 Add part

Clicking on add part adds a new square part to the drawing editor.

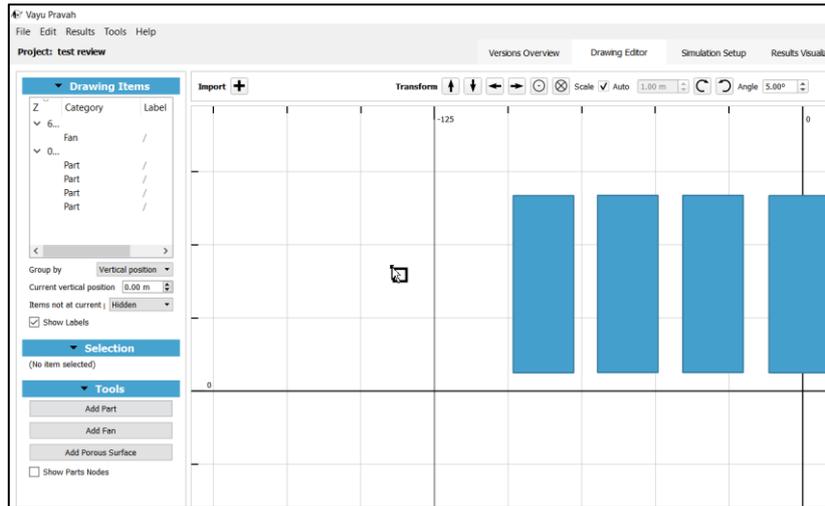


Figure 2-25 To add a part, click on add part, specify the position in drawing editor and click

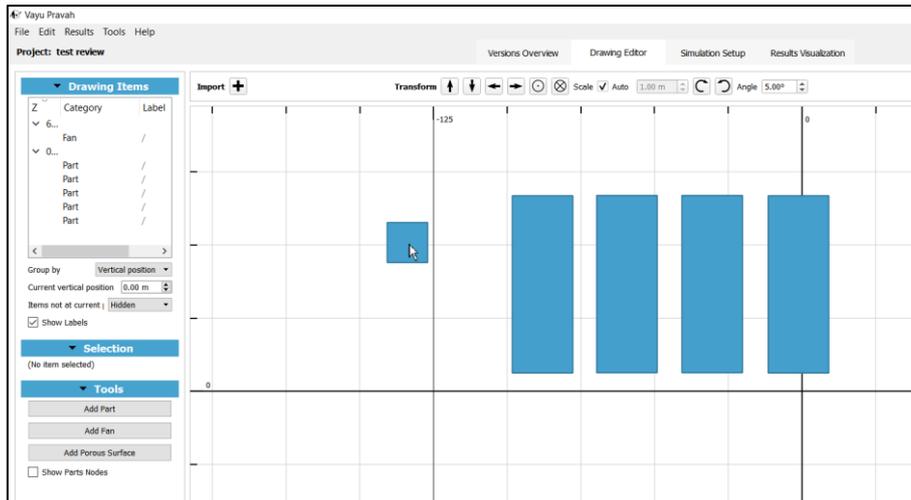


Figure 2-26 New part is added

2.2.5.2 Add fan

When the wind is too weak to generate cross ventilation, then it is possible to add an object which generates a pressure difference, like a fan which leads to very low energy mechanically driven air movement.

The user in this case has the facility to add a fan to the drawing by clicking on Add Fan. The user can specify the vertical position, the radius, and the horizontal and vertical angles of the fan in the selection section. The magnitude of wind through the fan can also be specified here.

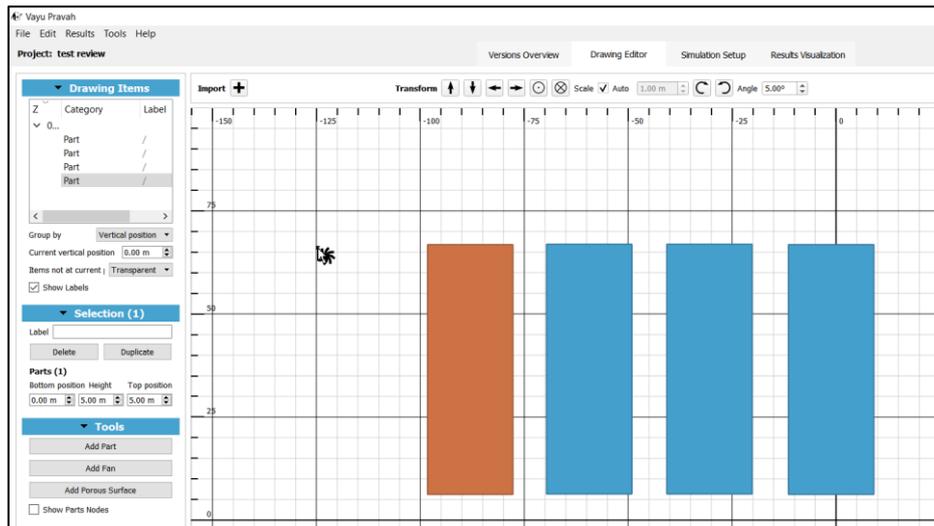


Figure 2-27 To add a fan, click on Add fan, specify the position in the Drawing editor and click

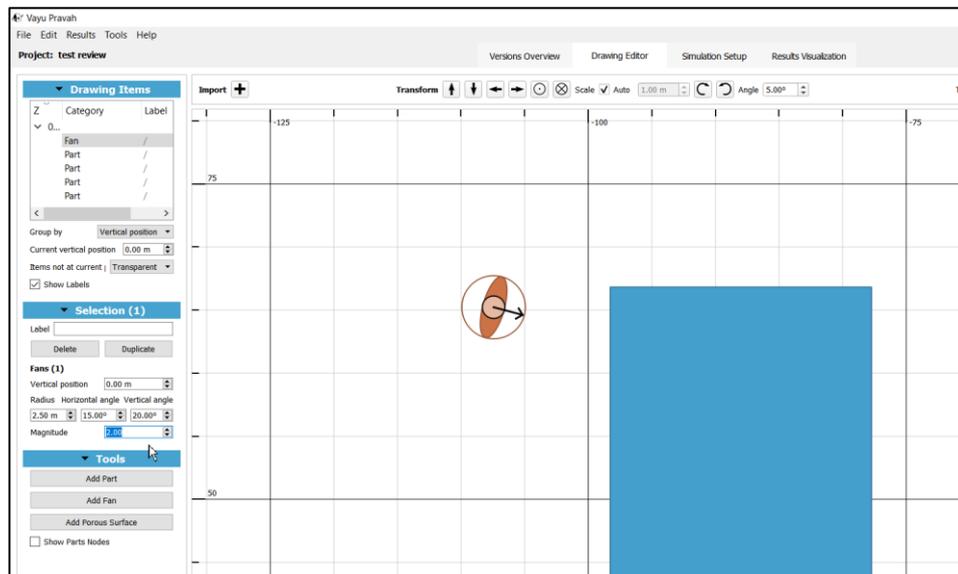


Figure 2-28 Once a fan has been added, its properties such as the radius, vertical and horizontal angle, and the magnitude can be added in the left-hand panel

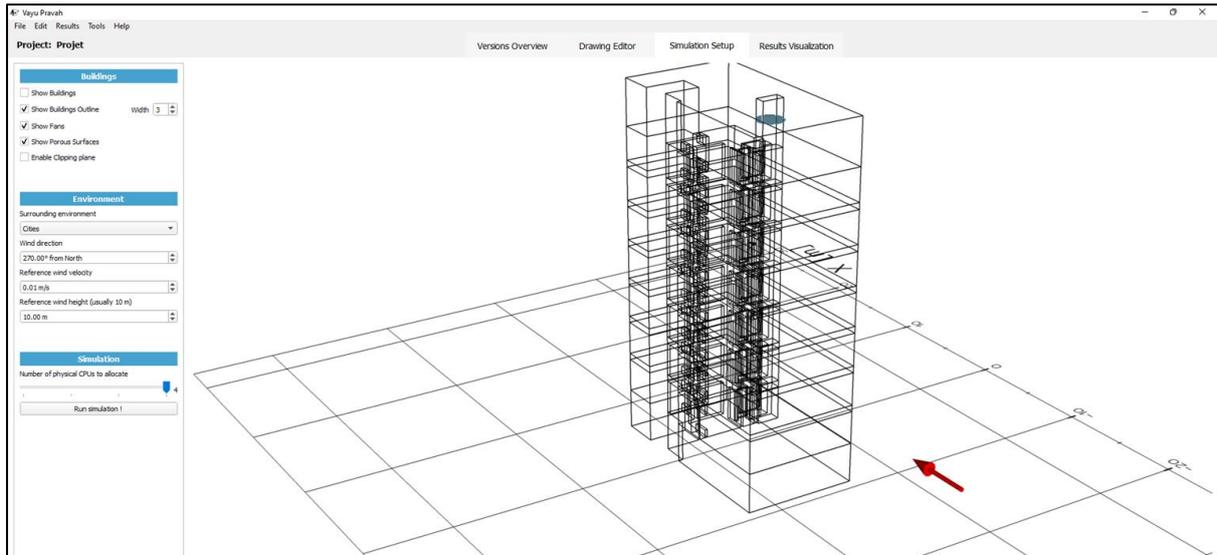


Figure 2-29 Simulation view with a fan added in the building

2.2.5.3 Add porous surface (resistance like a mosquito net)

This is a new feature that allows the user to add a porous surface at the desired part of the building.

To add a porous surface, In drawing editor, select the Add Porous surface option.

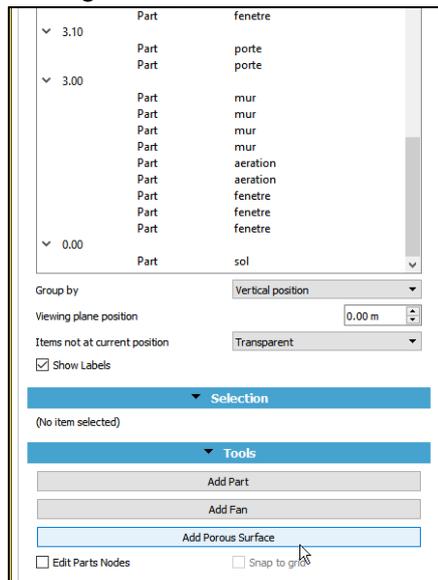


Figure 2-30 Click on Add Porous Surface to add a resistance

Move to the Drawing Editor and left click on where the surface should be added.

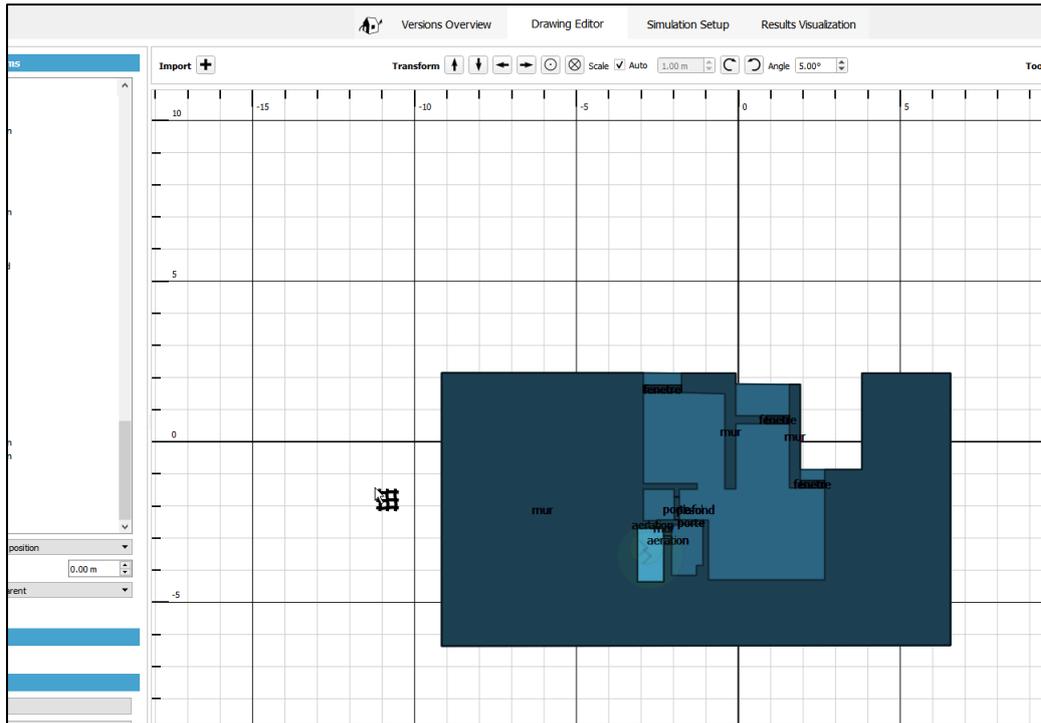


Figure 2-31 Placing the porous surface on the project in the Drawing Editor

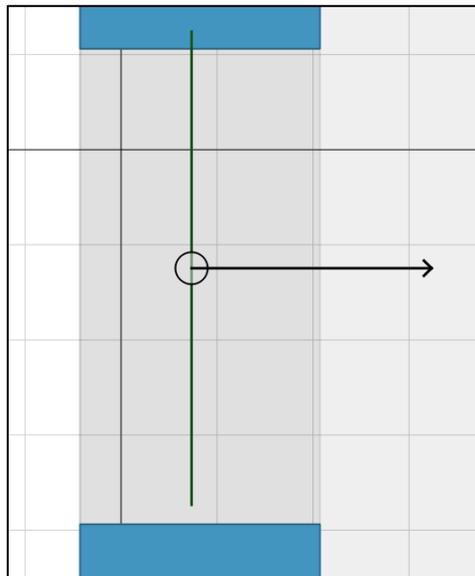
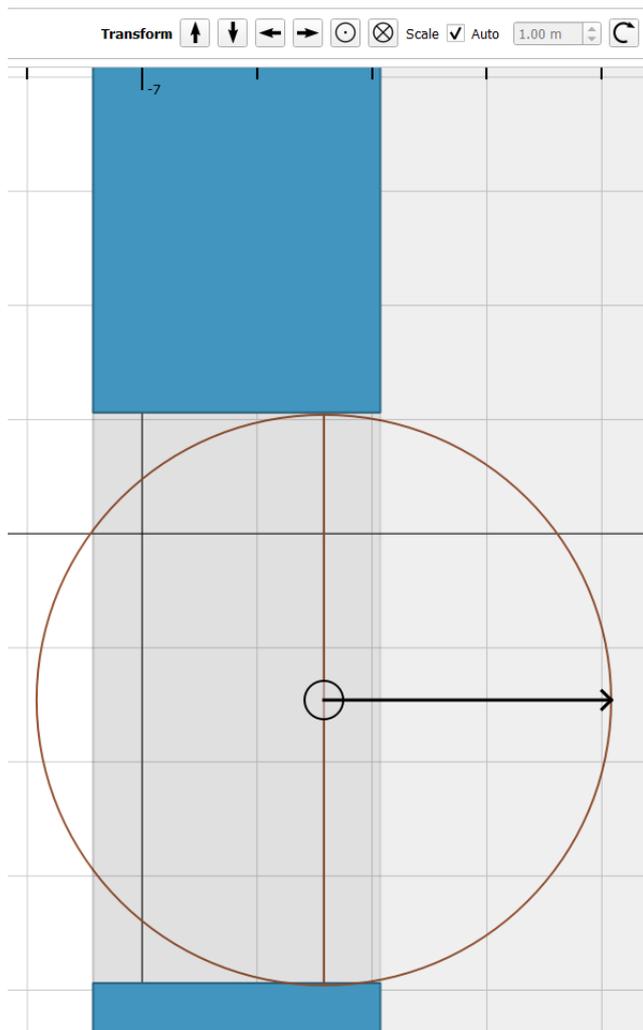


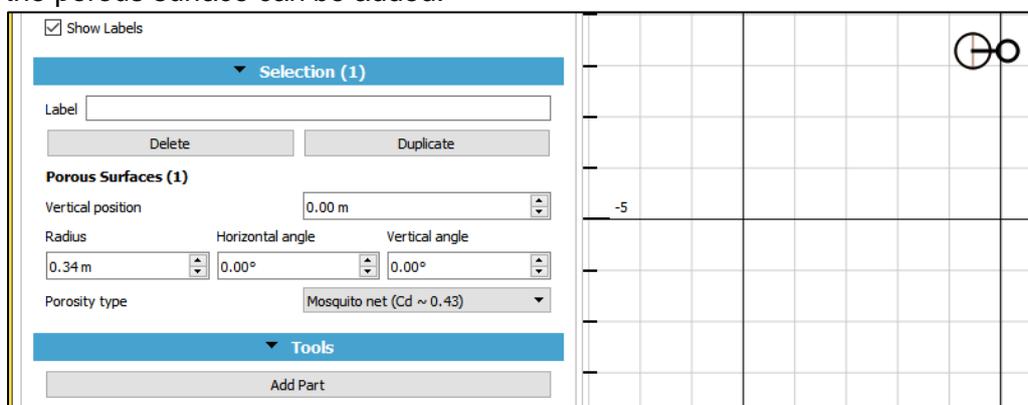
Figure 2-32 Porous surface added to the drawing editor

Then the user can left click on the center of the porous surface, and use the arrow buttons to move the surface around.



2.2.5.4 Defining the porosity parameters

Select the porous surface (either by clicking on it in the Drawing Editor or selecting it in the Drawing items menu on the left-hand side panel). Then on the LHS panel, the parameters of the porous surface can be added.



The user can define the vertical position, radius, angle horizontal and/or vertical angle of the porous surface in the selection panel on the left-hand side. The porosity of the surface can also be added here.

For adding the porosity, the user can choose from three different scenarios in the Porosity type drop down menu.

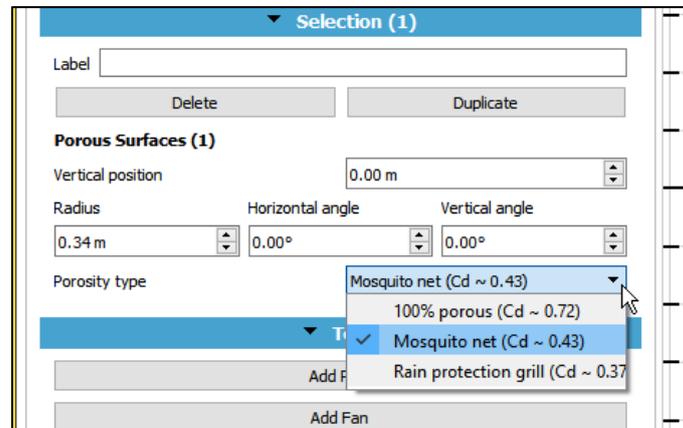
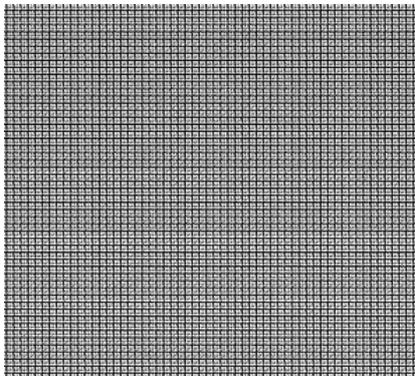


Figure 2-33 The user can choose between different values of porosity for the resistance

The value 100% porous corresponds to a 100% free opening (the value of $CD=0.72$ is only illustrative).

The mosquito net corresponds to a typical mosquito net, the discharge coefficient has been assumed to be 0.43 which corresponds to a polyester rectangular mesh with the yarn of 0.2 mm diameter and an opened area of 50%.



For a rain protection grille one has taken a grille which is protecting from the rain, but with large opened passage for natural ventilation ($CD=0.37$)



2.2.5.5 Editing parts

The user can edit the parts shape by using the Edit parts Nodes option. Selecting this option shows the nodes on the parts, along with their coordinates which can then be used to modify the shape of the parts.

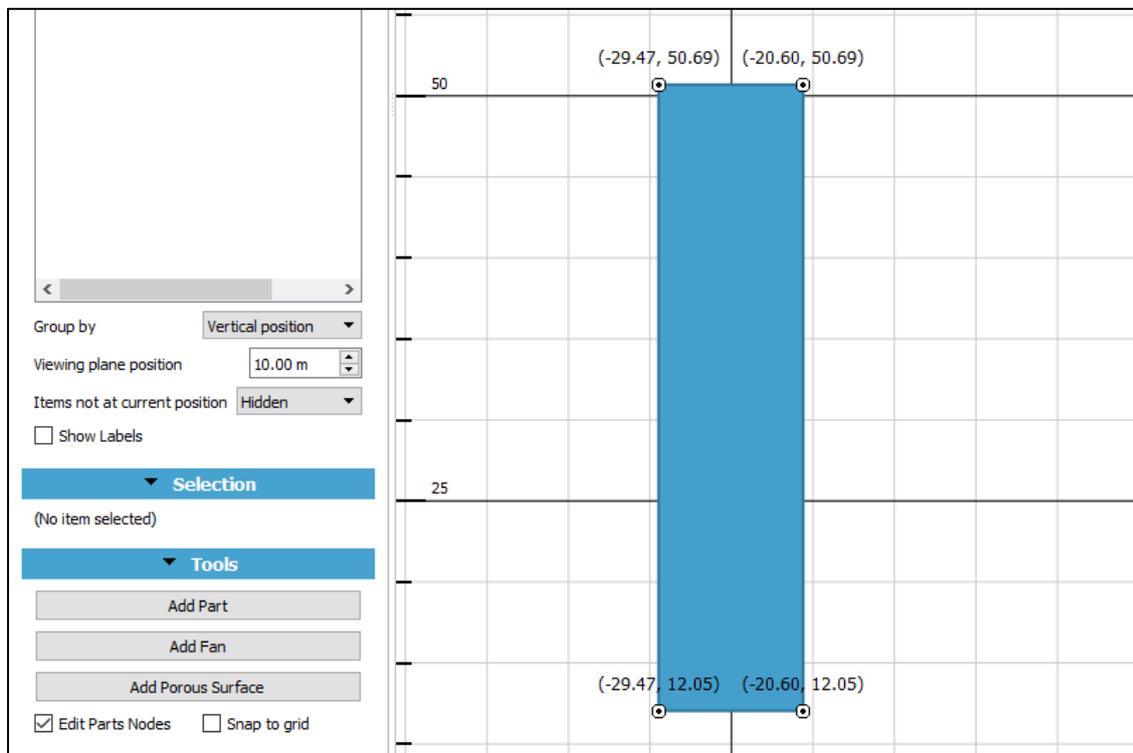


Figure 2-34 Checking the "Edit Parts Nodes" shows the nodes and their coordinates on the Drawing Editor. These can then be modified to modify the shape of the part

Also, to create a node anywhere on the part, the user can simply double click on the edge of the part (while Edit Parts Nodes is enabled), and a node along with its coordinates appears at this point.

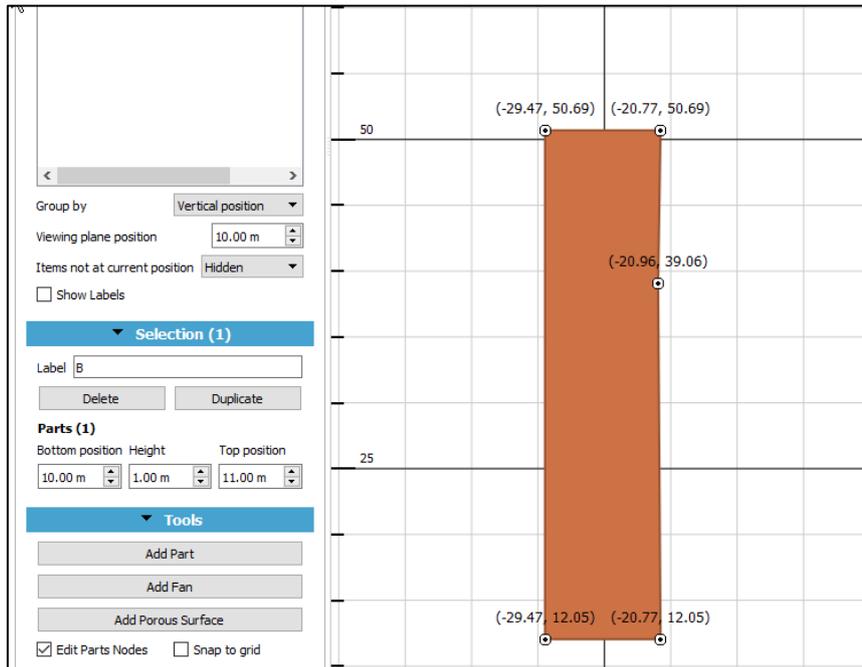


Figure 2-35 Creating a new node by double clicking on the edge of the part

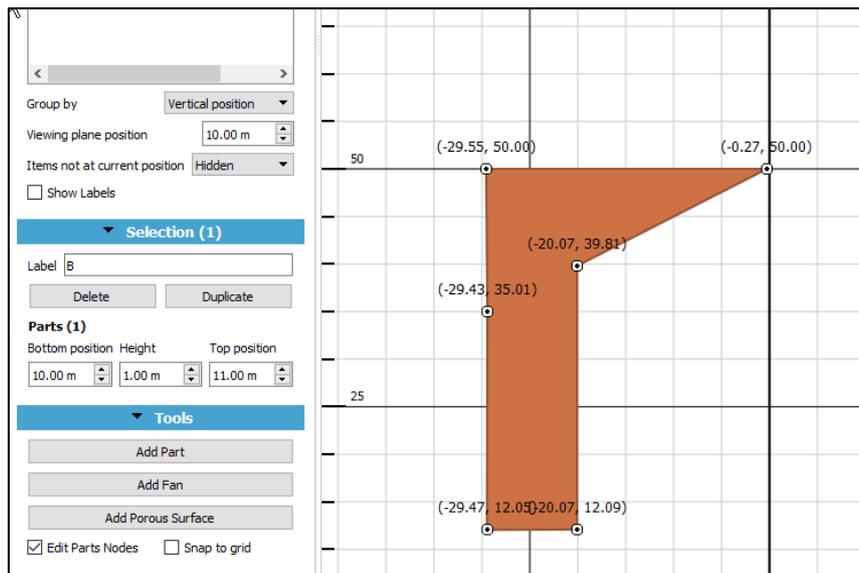


Figure 2-36 Changing the shape of the part by pulling the nodes

The Snap to Grid can be enabled to snap the nodes and align the part to the grid.

Also, the user can create holes in a part by double clicking inside the part while Edit Parts Nodes is enabled.

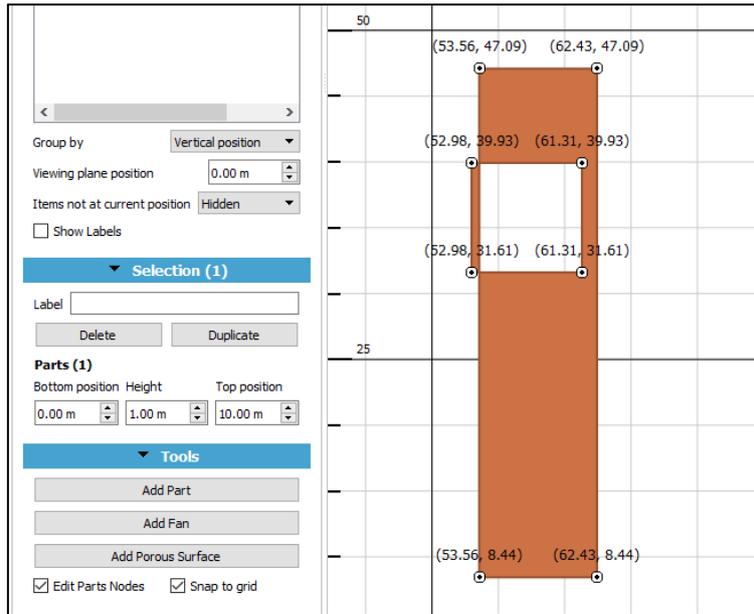


Figure 2-37 Creating a hole inside a part by double clicking inside it while Enable Parts Nodes is enabled

2.2.6 Transform functions

On the top panel in the drawing editor are certain functionalities that can help to transform the drawing. These can be applied to one part of the drawing or the entire drawing (by selecting the parts where transformation is desired). These are being discussed here :



Figure 2-38 Transform functionalities in Drawing Editor



2.2.6.1 Movement arrows

The movement arrow buttons can be used to move the drawing or a part of in the desired direction left, right, up or down.



2.2.6.2 Cross and dot buttons

The “cross” moves the selected items downwards, while the “dot” moves the selected items upwards. They are the 3D equivalent of the “Left”, “Right”, “Up”, and “Down” arrow buttons.

2.2.6.3 Scale and Auto functions

The scale and auto functions control the amount of distance moved by the movement arrows in a single click. If the user wants to change the unit moved for each click of the movement arrows, they can unselect the Scale, and enter a desired value in the field next to the Auto checkbox. The selected part shall then move by this distance on each movement.

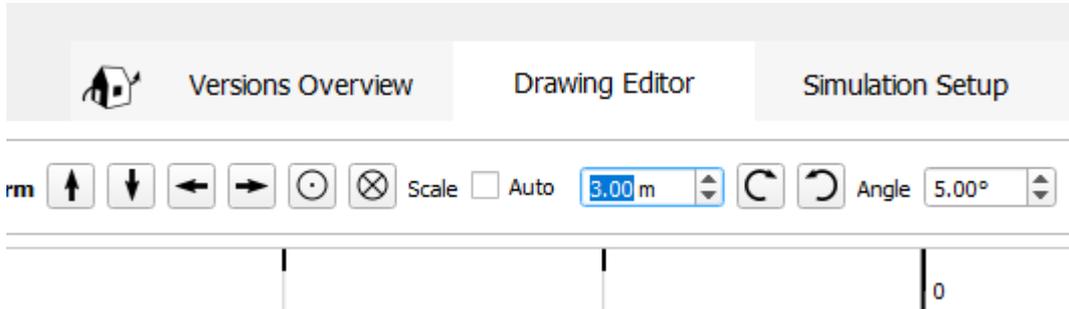


Figure 2-39 Scale and auto functions, uncheck the box next to Scale to control the distance moved by each click of the arrow key

2.2.6.4 Curved arrows

The curved arrows are used to rotate different parts in clockwise or anti-clockwise direction.

The user can also specify an angle by which the part should be moved and then click on either arrow to move the building.

2.2.6.5 Adjusting view

The view of the plan can be moved using the '+' icon. Click on the icon to select this mode. The view can then be moved by clicking and dragging inside the drawing view.

2.2.6.6 Selecting 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 planes, click on the '🏠' icon. This will select all the buildings in the highlighted area in all the planes.

2.3 Simulation Setup

Once a drawing has been completed in the Drawing Editor, the user can enter the Simulation Setup tab. Here the user can prepare the simulation case by entering building parameters such as wind direction, reference height, surroundings etc. The user can also visualize the 3D representation of the building(s).

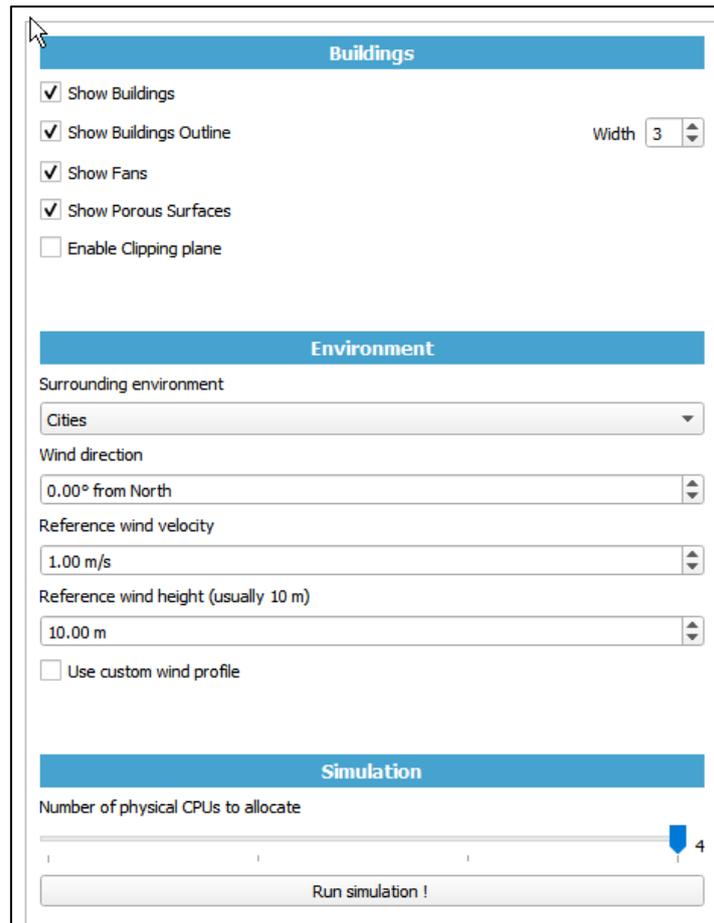


Figure 2-40 Defining the parameters in the simulation setup

2.3.1 Buildings

This section allows the user to visualize in different ways the 3D model of the complete building.

2.3.1.1 Show buildings

The user can choose to see the full buildings or just the outline by checking/unchecking the Show buildings option

2.3.1.2 Show buildings Outline

The user can choose to have an outline of desired width for the drawing. If no outline is desired, uncheck the Show Buildings Outline box.

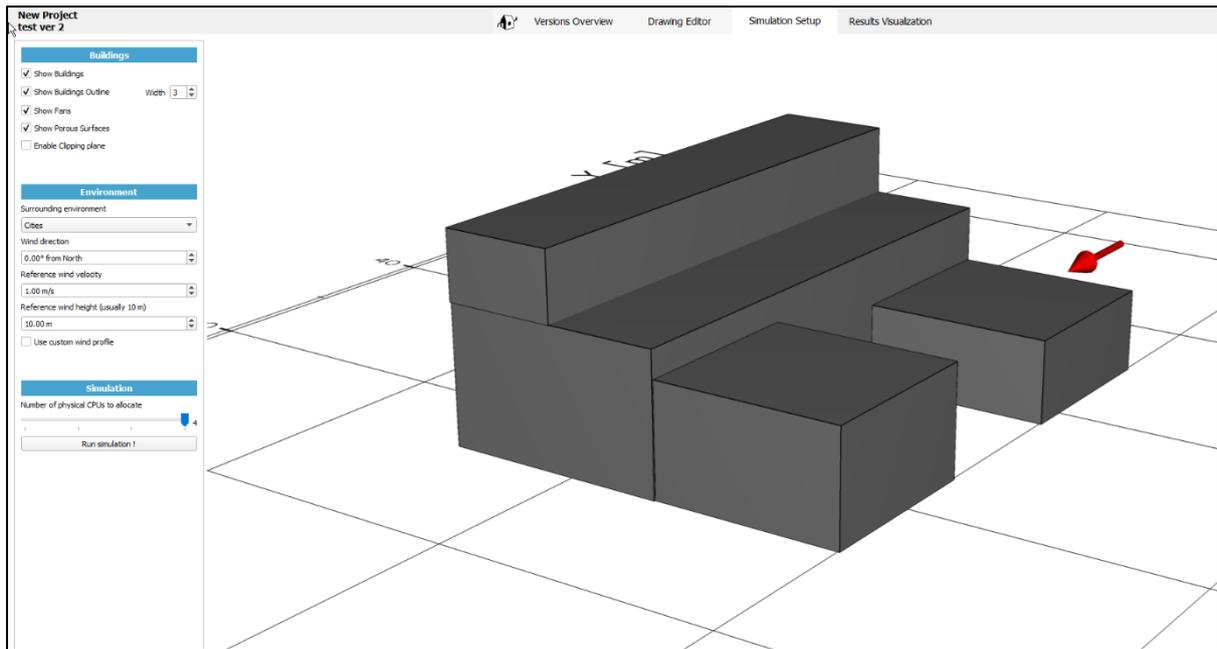


Figure 2-41 Viewing the 3D model of the building

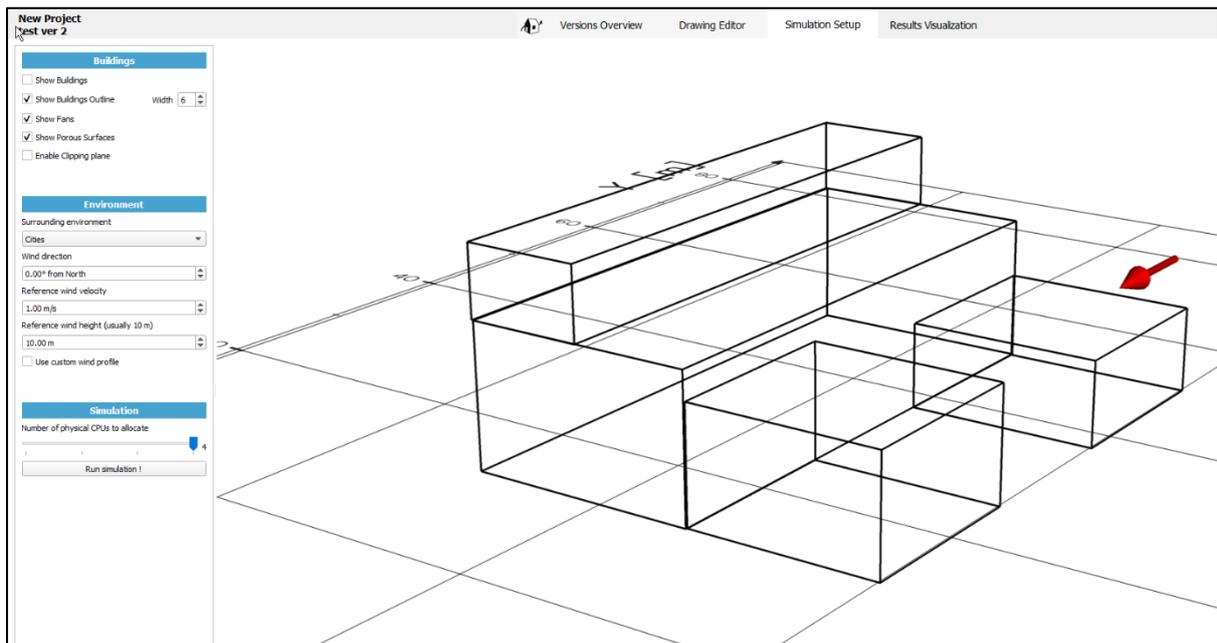
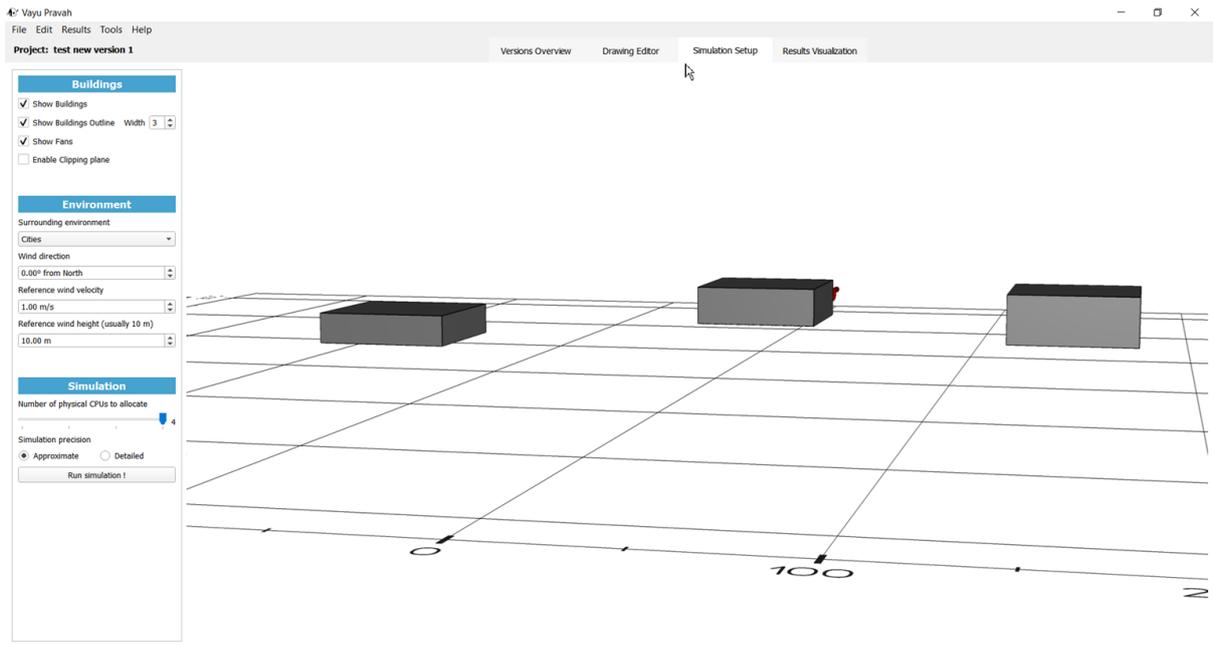


Figure 2-42 Viewing the 3D model of the building with just the Outlines

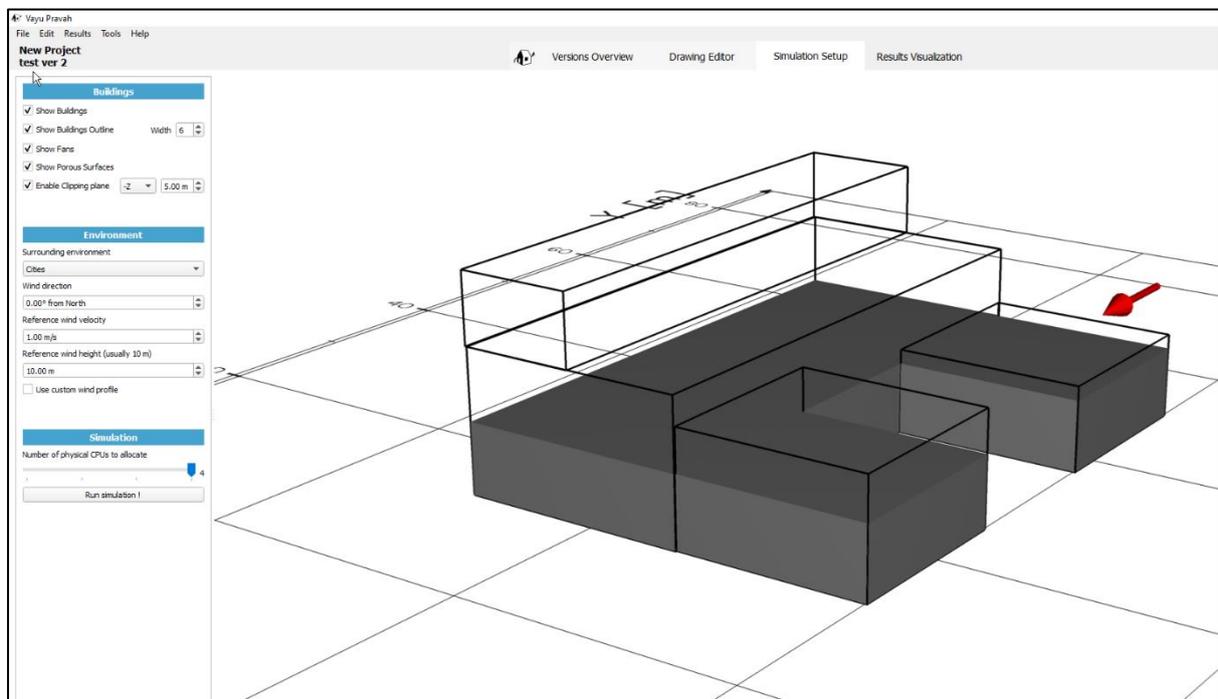
2.3.1.3 Show Fans and Show Porus Surface

Similarly, the user can choose to visualize the 3D model with and without fans or the porous surface by checking the respective boxes.



1.1.1.3 Enable Clipping pane

The user can click on the 'Enable Clipping Plane' to clip the 3 D view along a specified axis. The plane along which the view must be clipped, and the height from the bottom at which the clip is desired, can be specified in the fields.

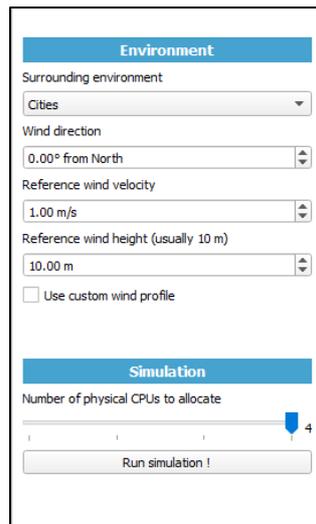


2.3.2 Prepare the simulation: Environment

The version of the case is then prepared for simulation by defining the environment and the wind conditions for the simulation.

2.3.2.1 Choose surrounding environment

The surroundings of the building massing (wind profile typology) can be chosen from the drop-down menu (cities, village etc.). The user can then specify the wind direction, velocity and the reference wind height.



The screenshot shows a software interface for configuring simulation parameters. It is divided into two main sections: 'Environment' and 'Simulation'.
Under the 'Environment' section, there are four dropdown menus: 'Surrounding environment' (set to 'Cities'), 'Wind direction' (set to '0.00° from North'), 'Reference wind velocity' (set to '1.00 m/s'), and 'Reference wind height (usually 10 m)' (set to '10.00 m'). Below these is a checkbox labeled 'Use custom wind profile' which is currently unchecked.
Under the 'Simulation' section, there is a slider control for 'Number of physical CPUs to allocate' with a blue handle positioned at the value '4'. At the bottom of the panel is a button labeled 'Run simulation !'.

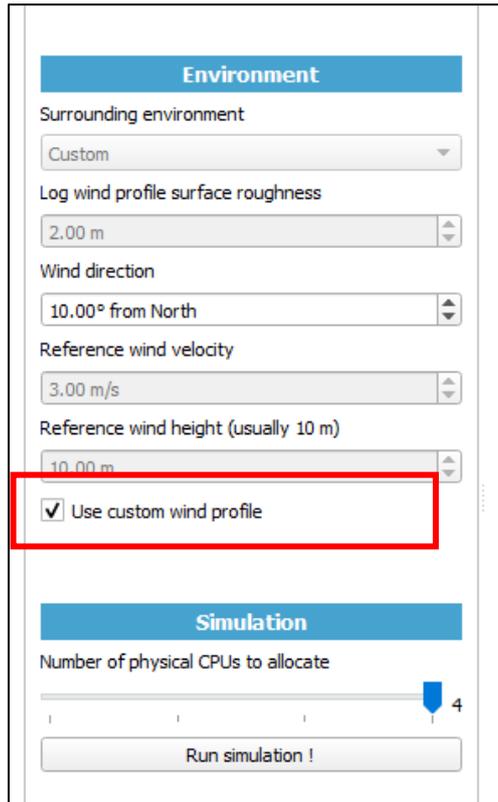
Figure 2-43 Defining the simulation environment

2.3.2.2 Choose the wind profile

Wind profile can be defined by choosing the wind direction (angle from North), and entering reference wind velocity and reference wind height. This will be reflected in the 3D model.

2.3.2.3 Custom wind profile

The user has the choice to enter a custom wind profile by clicking on the “Use custom wind profile” box on the left-hand side panel.



Once this option has been selected, on clicking Run simulation, the user will be prompted to enter the custom wind profile. As shown in the figure below, the user can also import the wind profile from an external source.

In the dialogue box, the user can add the different velocity components at different heights and create a custom wind profile.

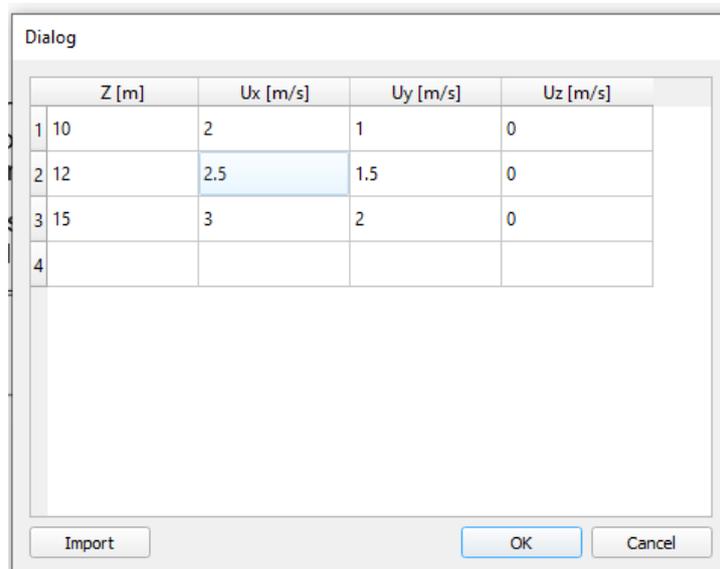


Figure 2-44 Entering a custom wind profile

2.3.2.4 Chose the number of CPU to allocate to the simulation

Normally, a significant reduction in simulation duration is obtained by choosing as many physical cores as available on the machine.

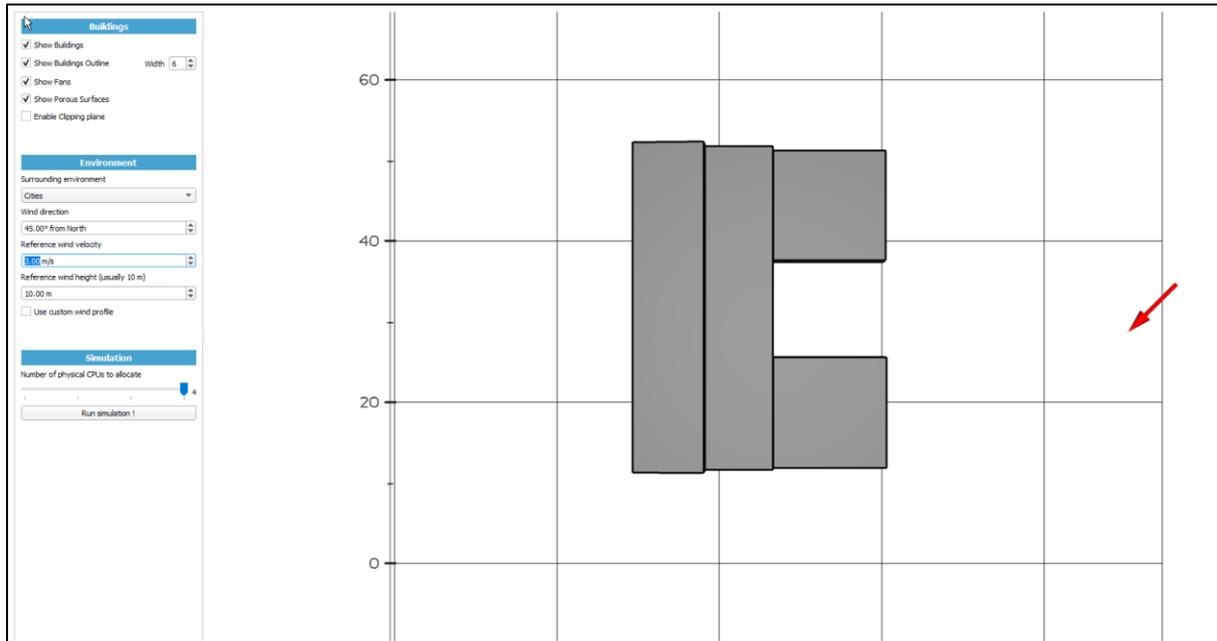


Figure 2-45 3D model ready for simulation

2.3.3 During the simulation

A pop-up window will appear indicating the status of the simulation. The 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.

2.3.3.1 Simulation running

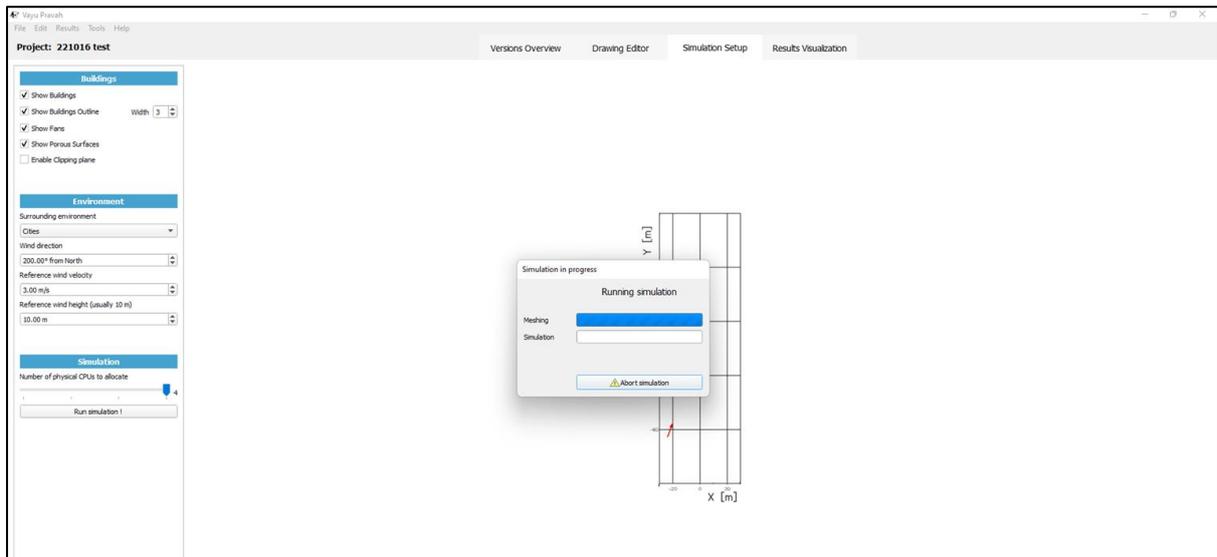


Figure 2-46 Simulation running with estimation of remaining time (after meshing)

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

2.3.3.2 The simulation is completed

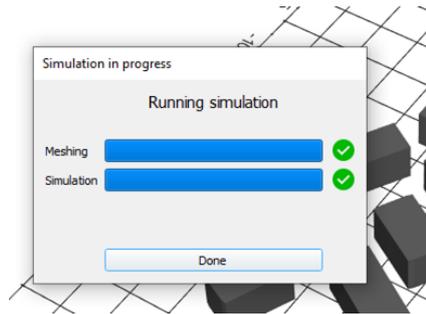


Figure 2-47 Simulation completed

The user can run multiple simulations in Vayu Pravah simultaneously in separate windows.

2.4 Results Visualization

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

2.4.1 Select the Results Visualization pane

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

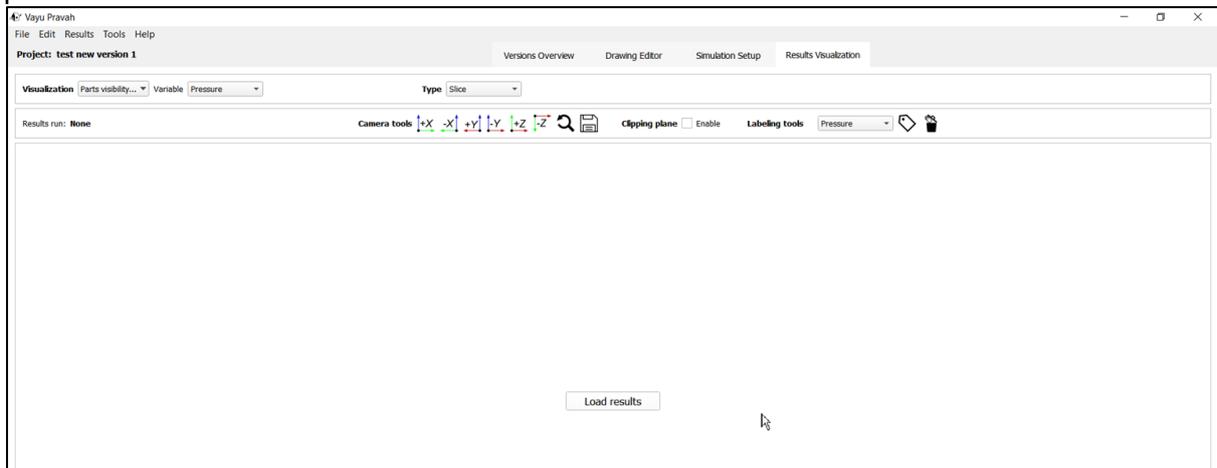


Figure 2-48 The Results Visualization pane

2.4.1.1 Select the version simulation result to load

To choose a previously simulated version to load on the results visualization pane, click on Results on the top left-hand panel, and click on Open Results. A window appears with all the versions from the cases from the current project for which simulation results are available. The user can here select the case for which simulation results are to be viewed. The '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.

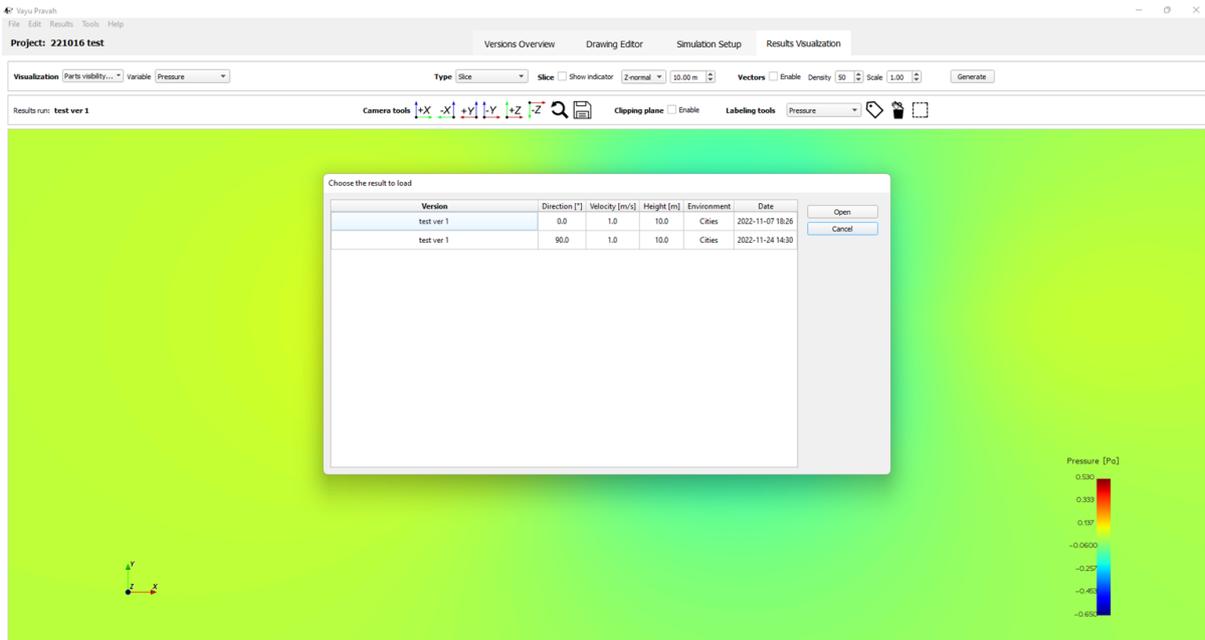


Figure 2-49 Selecting the case for which simulation results are to be viewed

1.1.1.4 Defining parts visibility

User can choose which parts of the drawing should be visible in the simulation. They can choose to see or hide the buildings, outlines, fans and porous surface, by choosing from the drop-down menu on the top left.



Figure 2-50 Choosing from the drop down menu the parts to be visible in the simulation

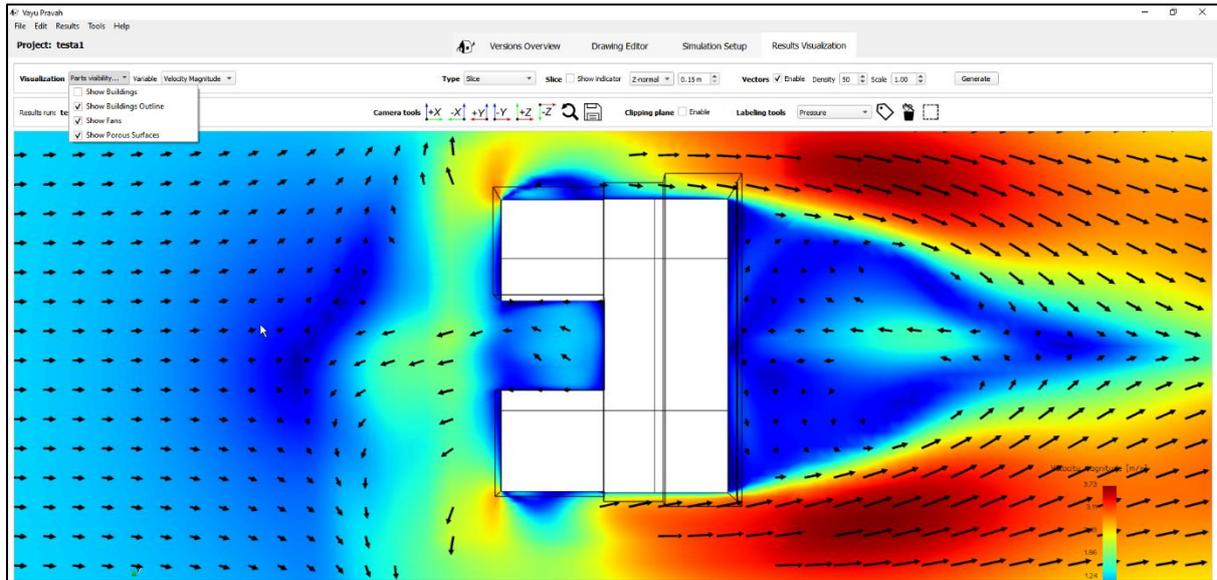


Figure 2-51 Selecting the parts of the building to be visible in the results

2.4.2 Selecting, preparing, and adjusting the View

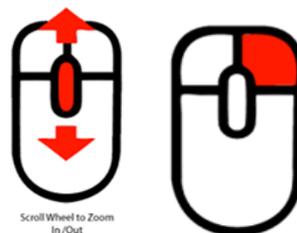
2.4.2.1 The different options of variables

One can select from five different parameters to visualize in the version simulation. These can be chosen 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))

2.4.2.2 Zoom and un-zoom

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

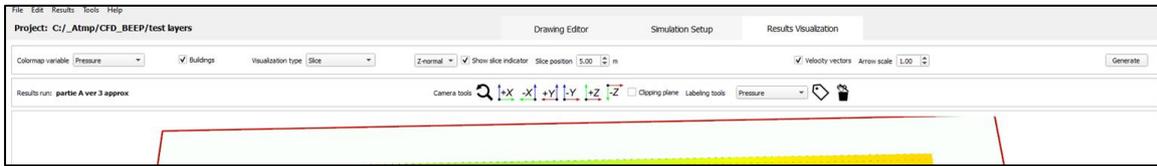


Zooming is obtained also by moving the mouse with the right button kept pushed (by sliding the mouse toward the screen)

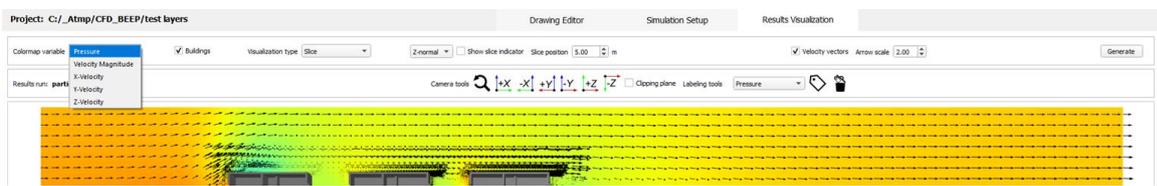
2.4.2.3 Panning with the cursor

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.



2.4.3 Selecting the kind of results presentation

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

2.4.3.1 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.

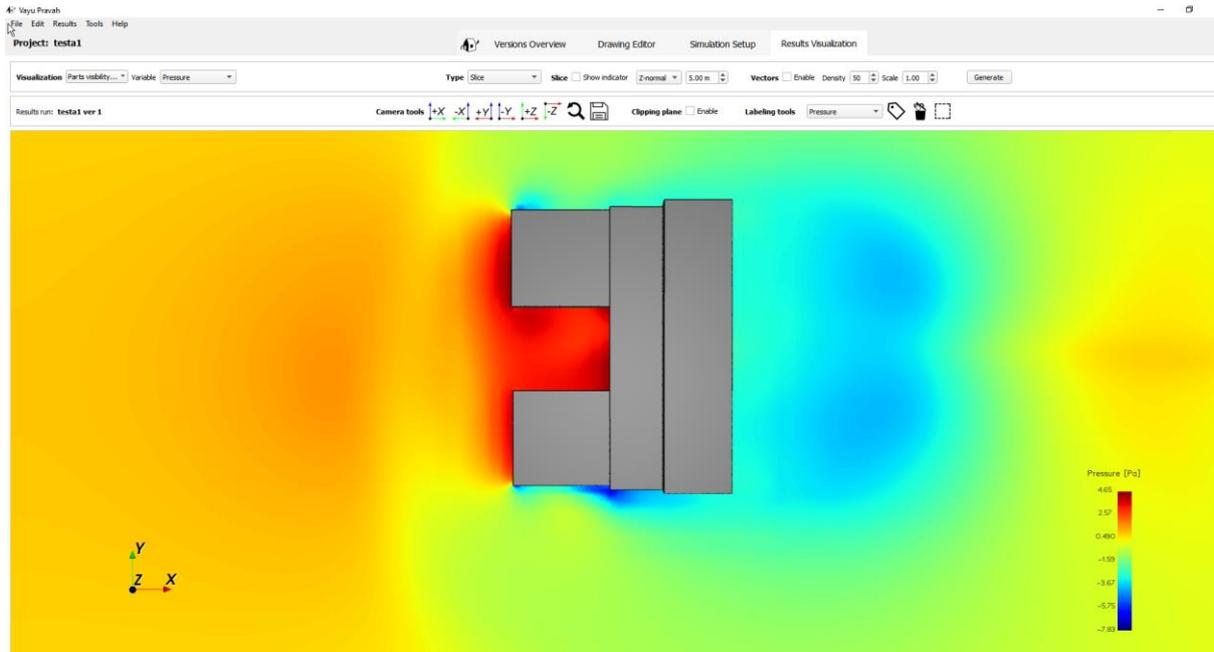


Figure 2-52 Simulation slice result (without slice indicator)

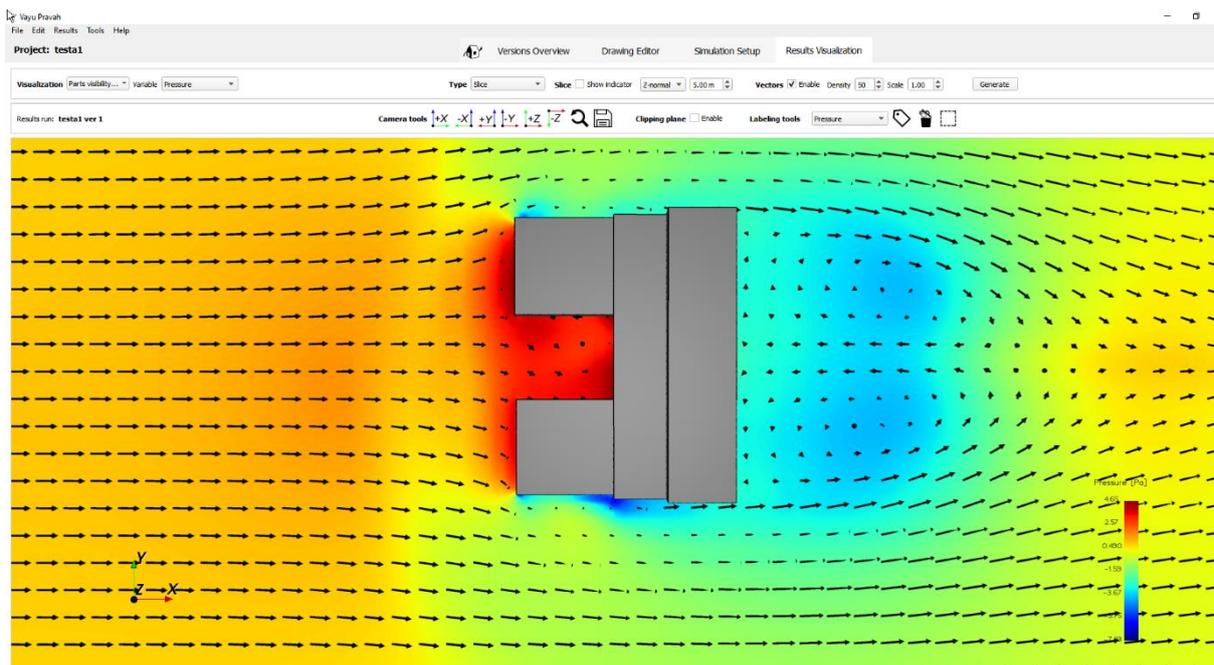


Figure 2-53 Slice visualization with vectors

Velocity vectors

In the slice visualization mode, one can show the velocity vectors by checking velocity vectors box. The density of the vectors and the scale size of the arrows can also be adjusted. This adds to better visualization of the airflow movement in the results.

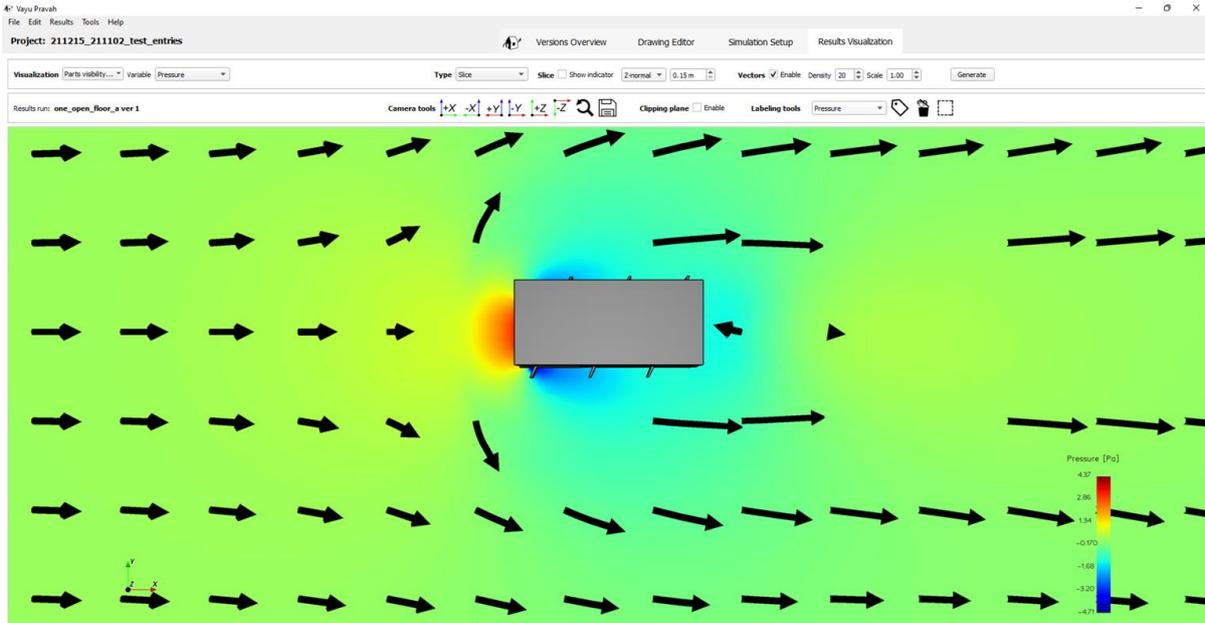


Figure 2-54 Velocity vectors are enabled, with low density (20)

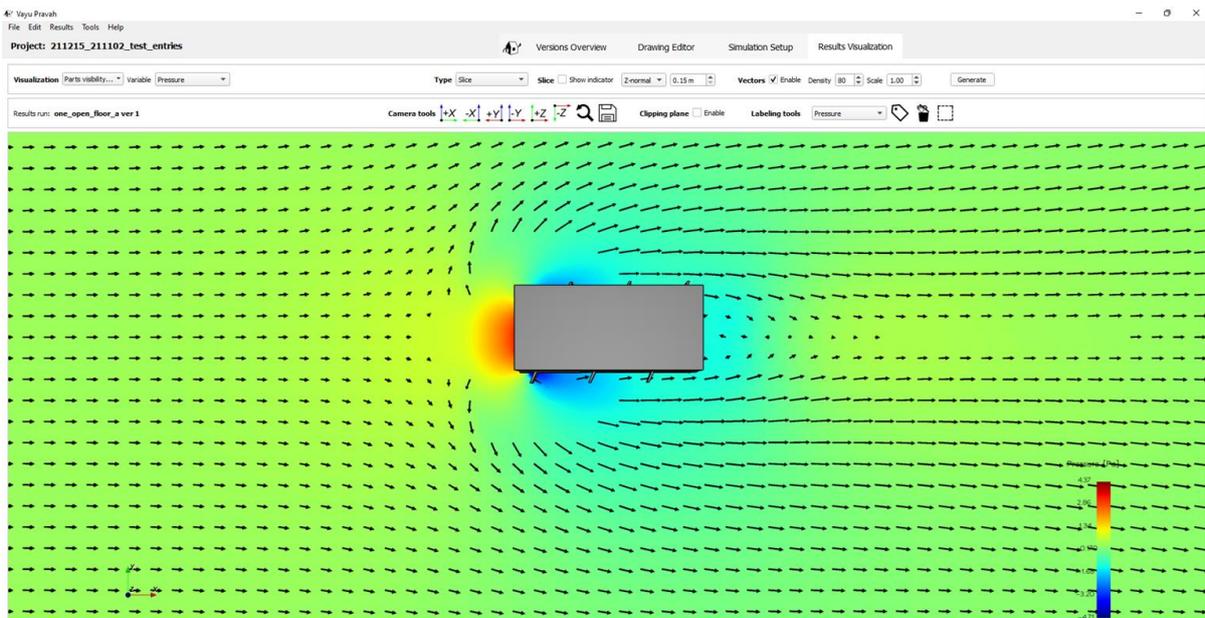


Figure 2-55 Velocity vectors are enabled, with high density (80)

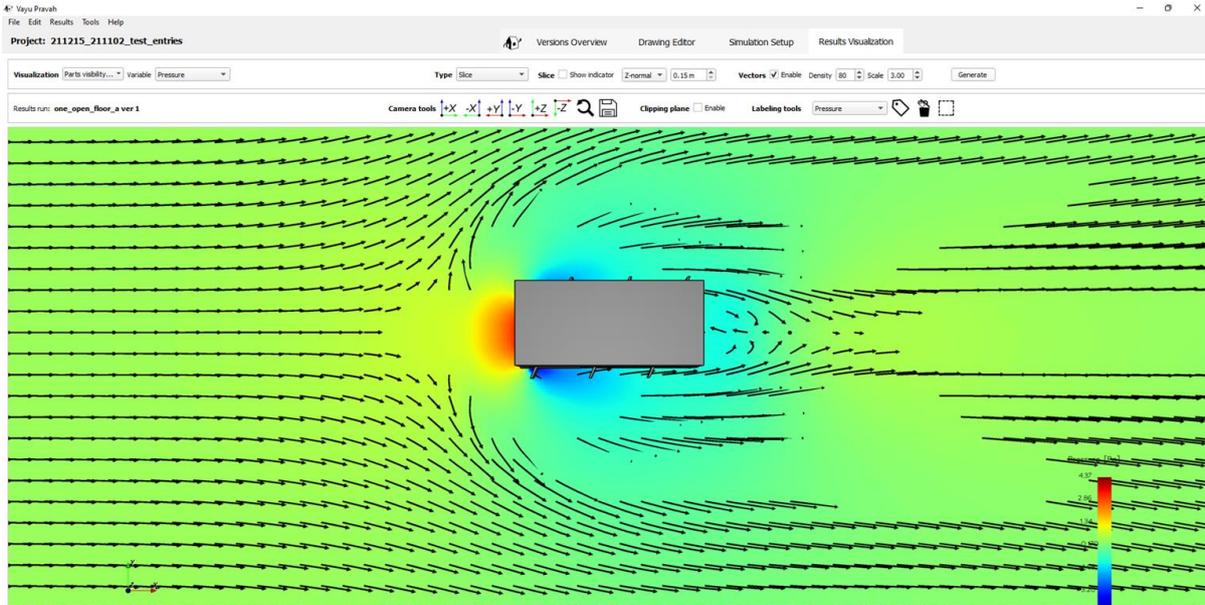


Figure 2-56 Velocity vector scale high (3)

2.4.3.2 Threshold

This representation shows the volume in which the value of the threshold variable is 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.

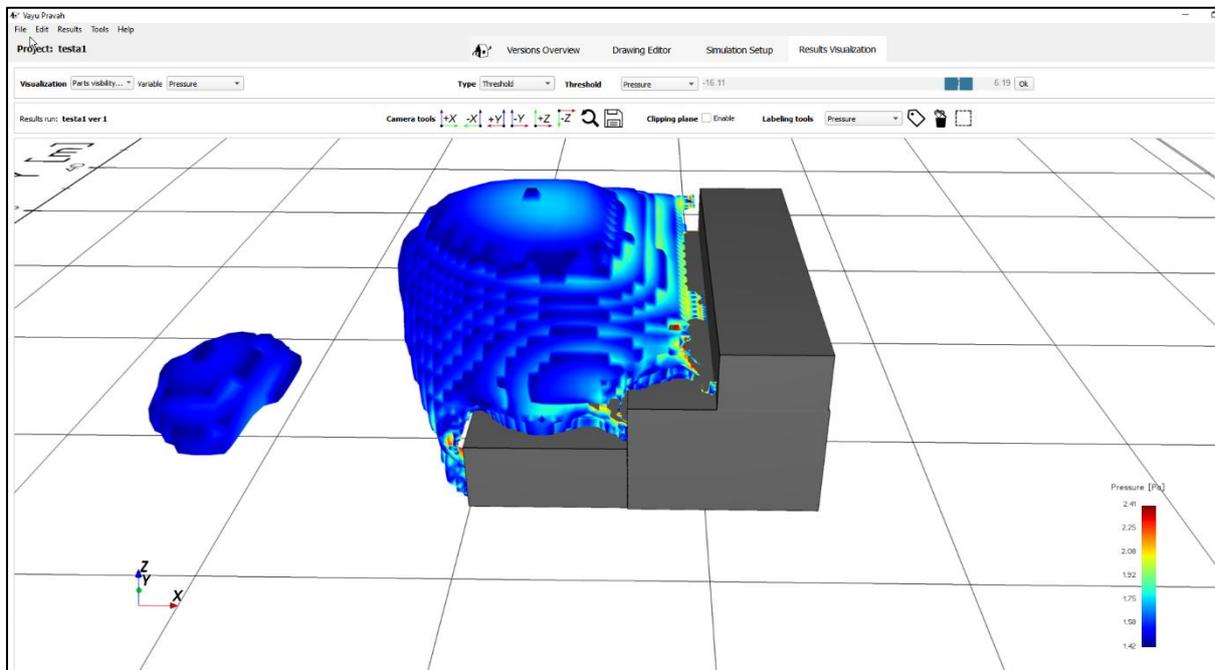


Figure 2-57 Threshold type visualization of pressure



Figure 2-58 Selecting the parameters, variable and the range for threshold visualisation

2.4.3.3 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'.

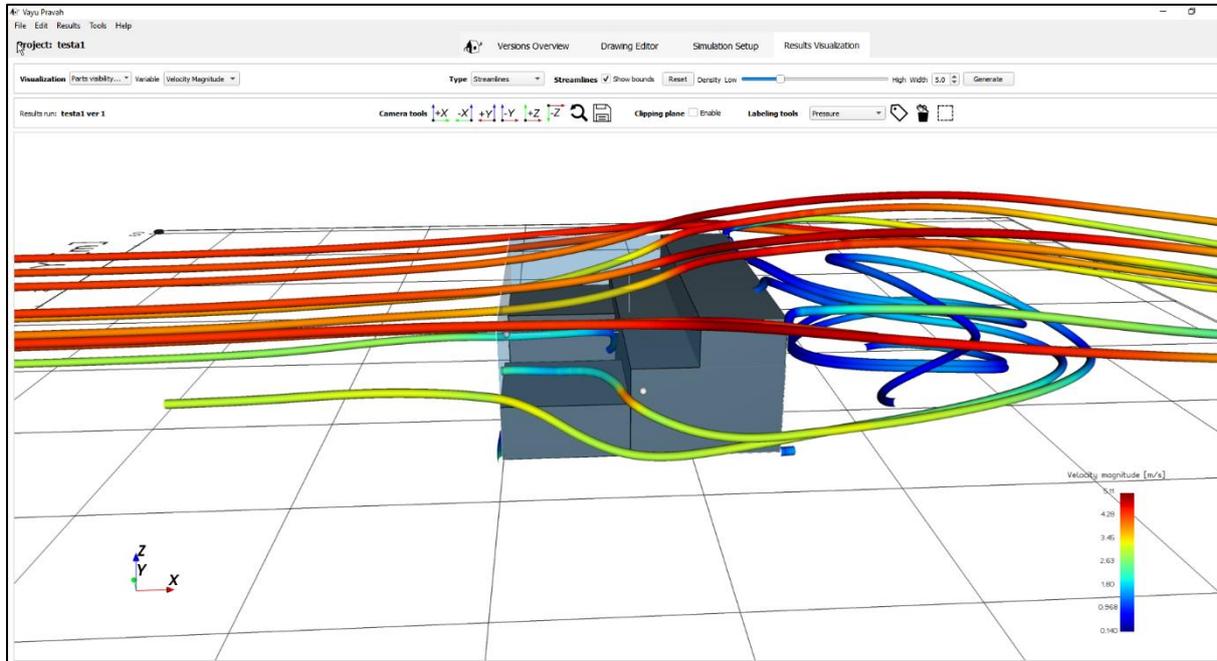


Figure 2-59 Streamline visualization

One can choose 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. The thickness of the streamlines can be varied in the box "Width".

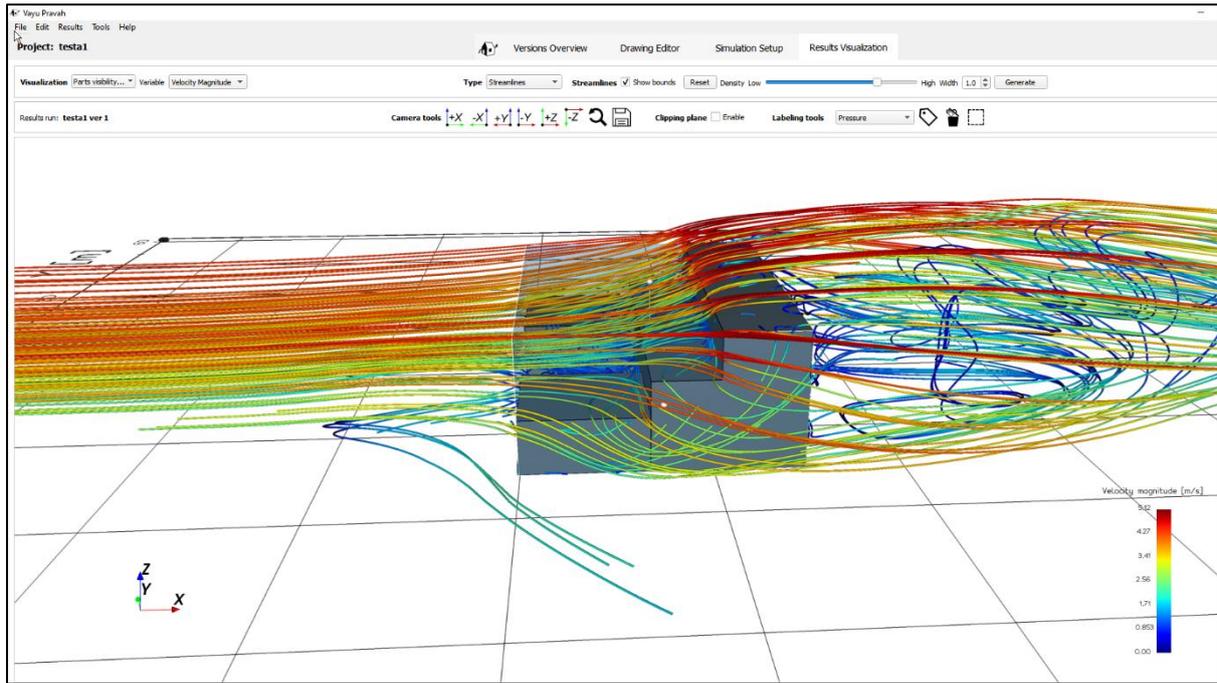


Figure 2-60 Streamline visualisation at high density of streamlines

2.4.3.4 Building surfaces

This visualization displays the buildings with their façade colored by either one of the five 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.

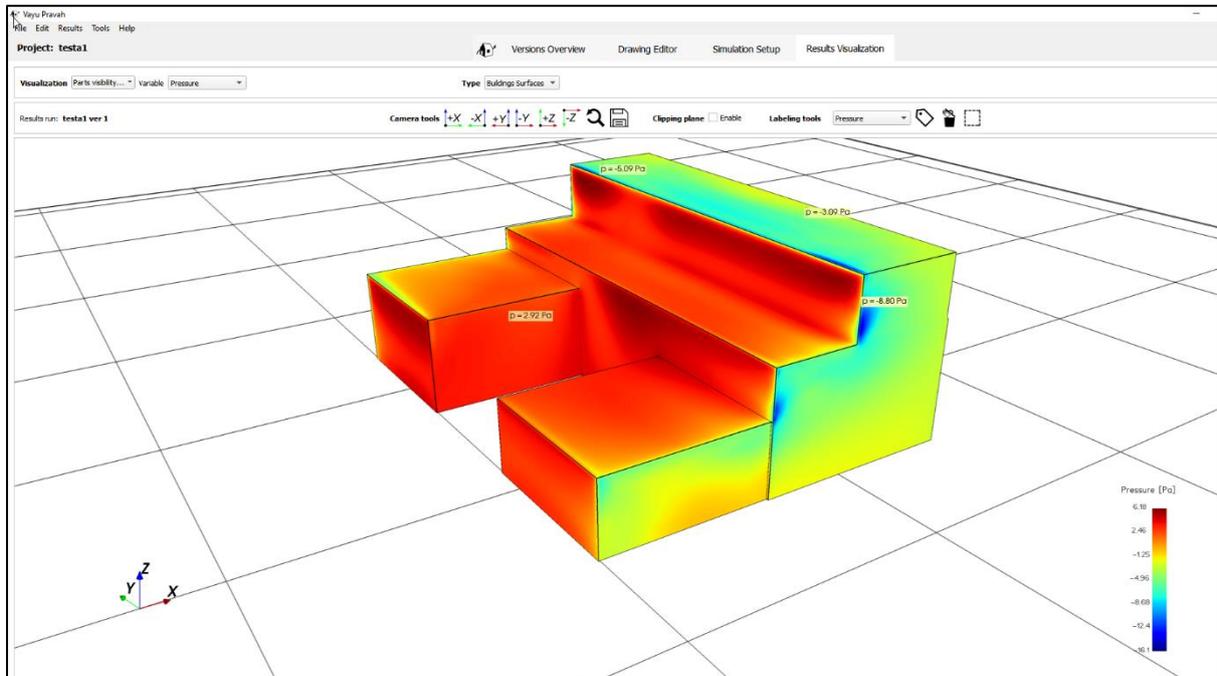


Figure 2-61 Building surface visualization, for viewing parameters at the building facade

2.4.3.5 Labels

Choosing the labels variables

Each 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))

Placing labels

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

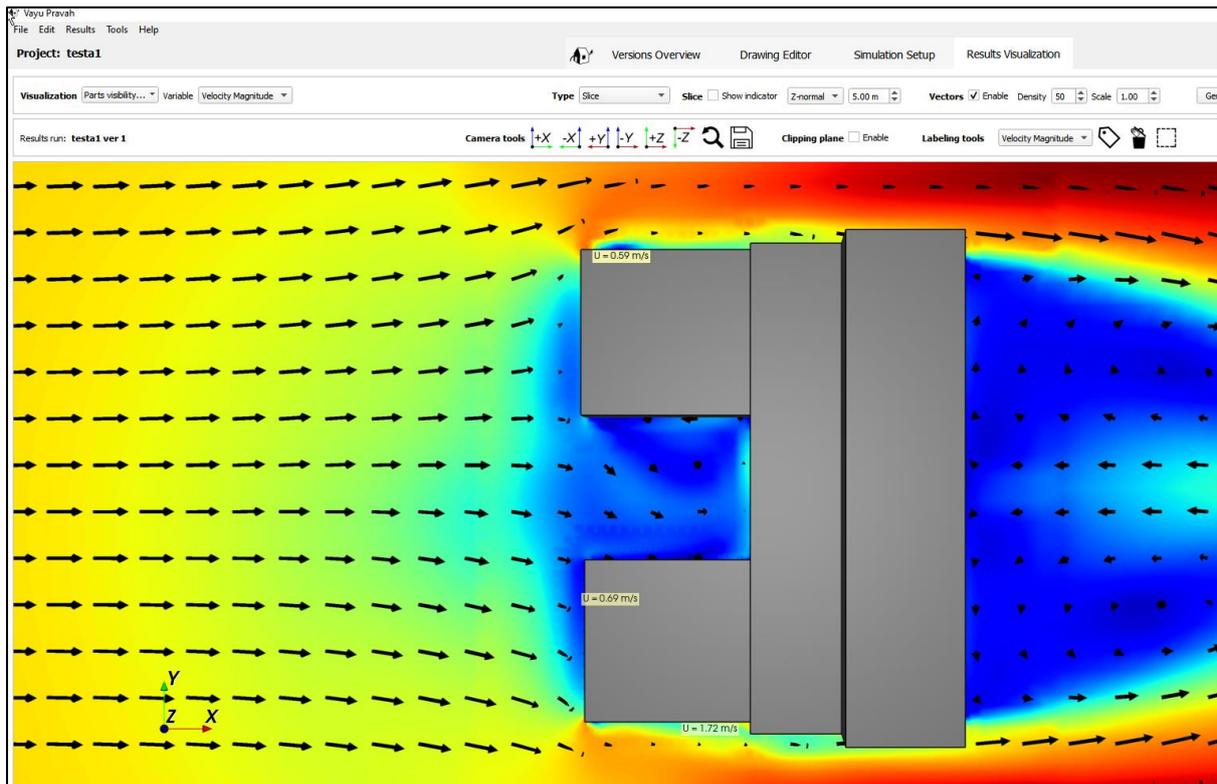


Figure 2-62 Results with velocity magnitude labels added

The user can also add labels from a parameter (variable like pressure, velocity etc..) different from the one being viewed. For example, in the figure below the pressure variable labels are added in a velocity magnitude visualization.

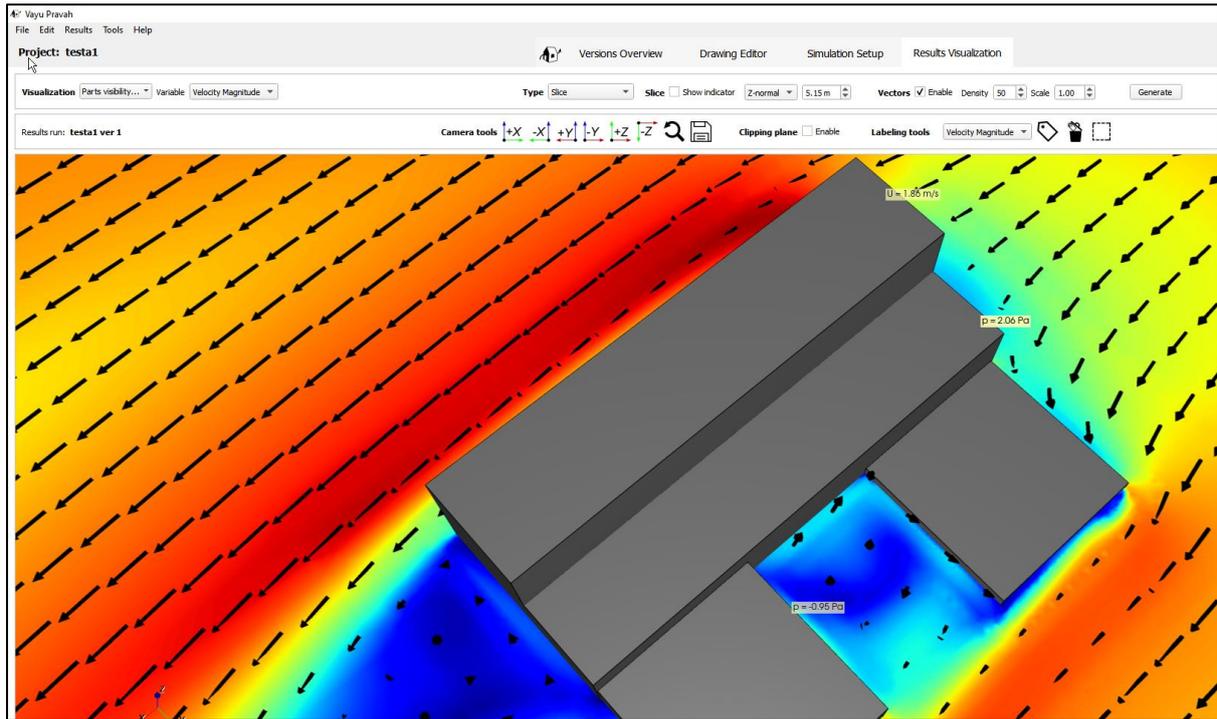


Figure 2-63 Adding pressure labels in velocity magnitude visualisation

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, click 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.

2.4.3.6 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

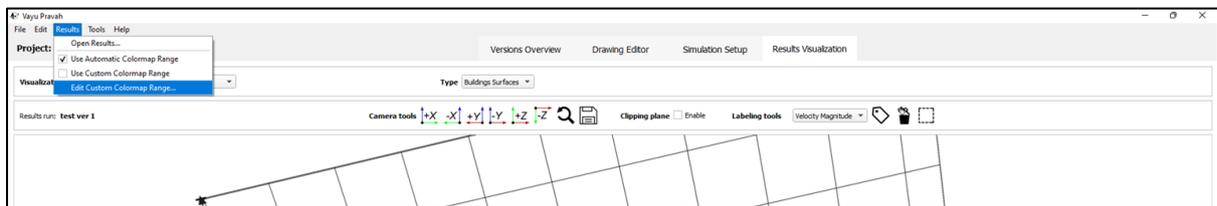


Figure 2-64 Adjusting color scale

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

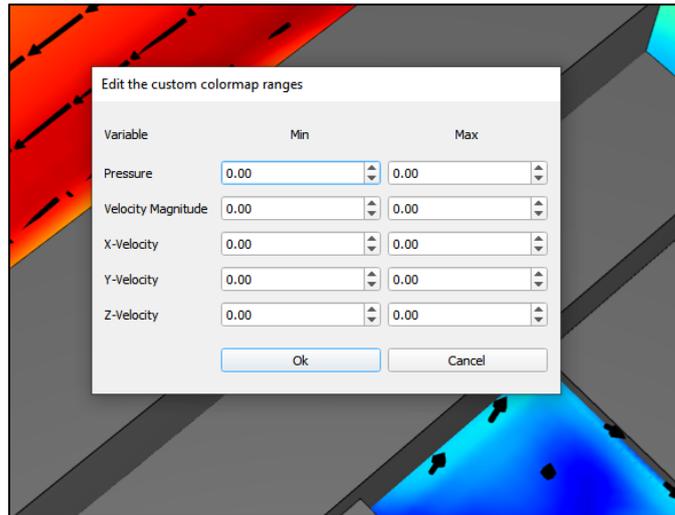


Figure 2-65 Adjusting the variable range

Click Ok to validate.

2.4.4 Visualization type

2.4.4.1 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.

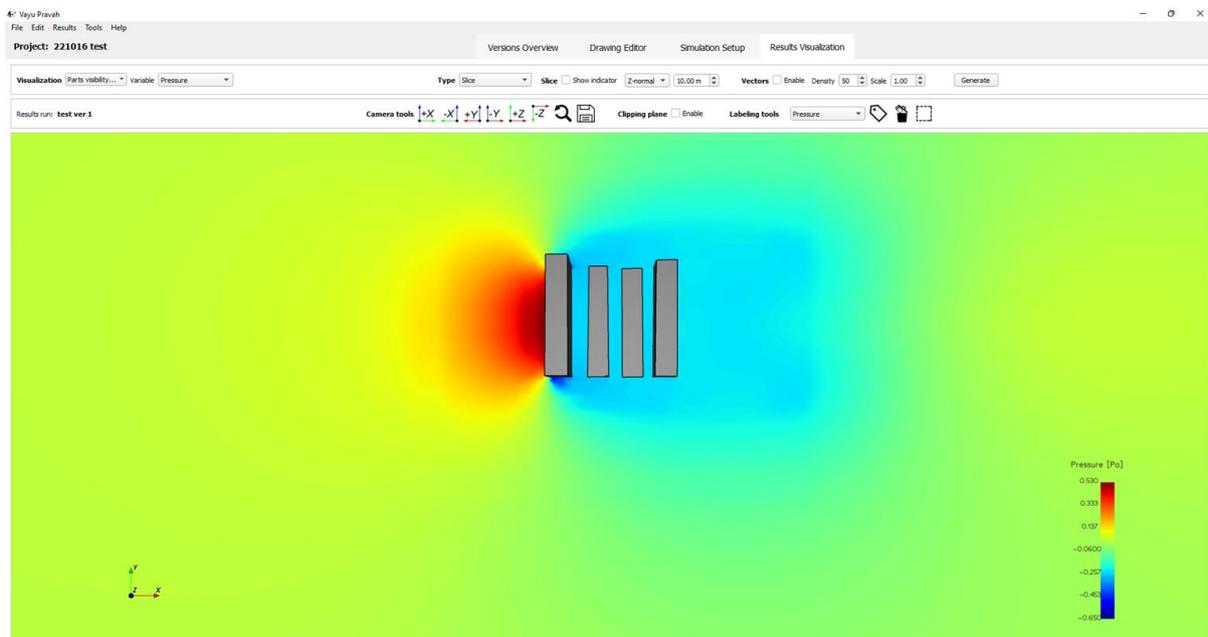


Figure 2-66 Visualizing pressure variable

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

2.4.4.2 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.

2.4.5 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.

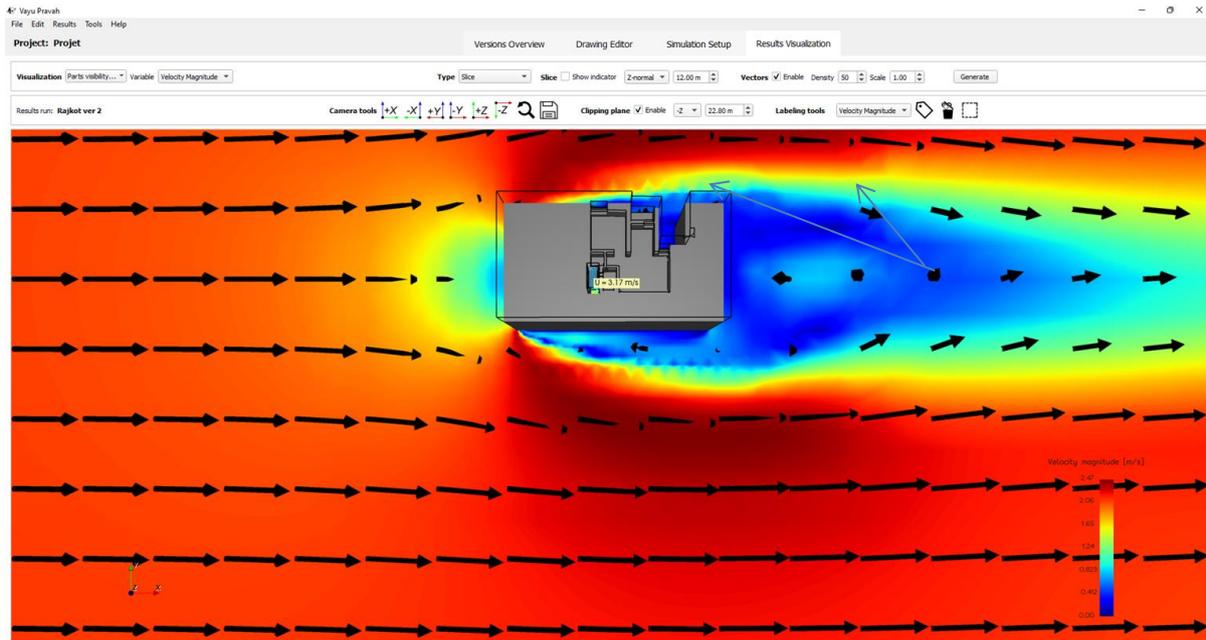
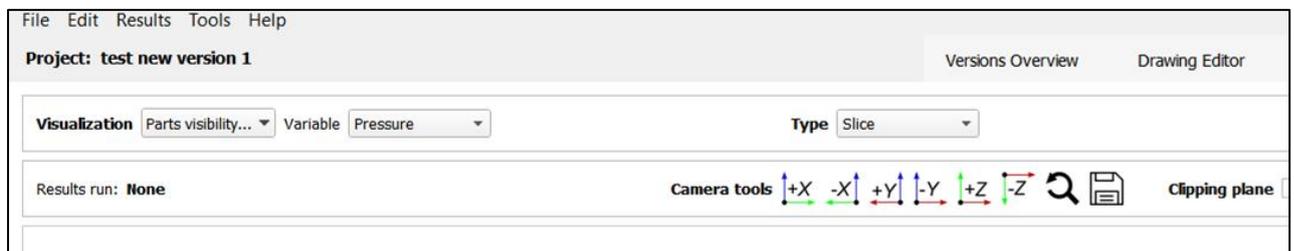


Figure 2-67 Clipped plane on Z axis

2.4.6 Taking Screenshot

User can take a screenshot of the results screen by clicking on the  icon. The screenshot taken is saved in the project folder.



2.4.7 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 the same parameters (range of pressure, velocity etc..) for comparing different cases.

2.4.7.1 Reload results

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

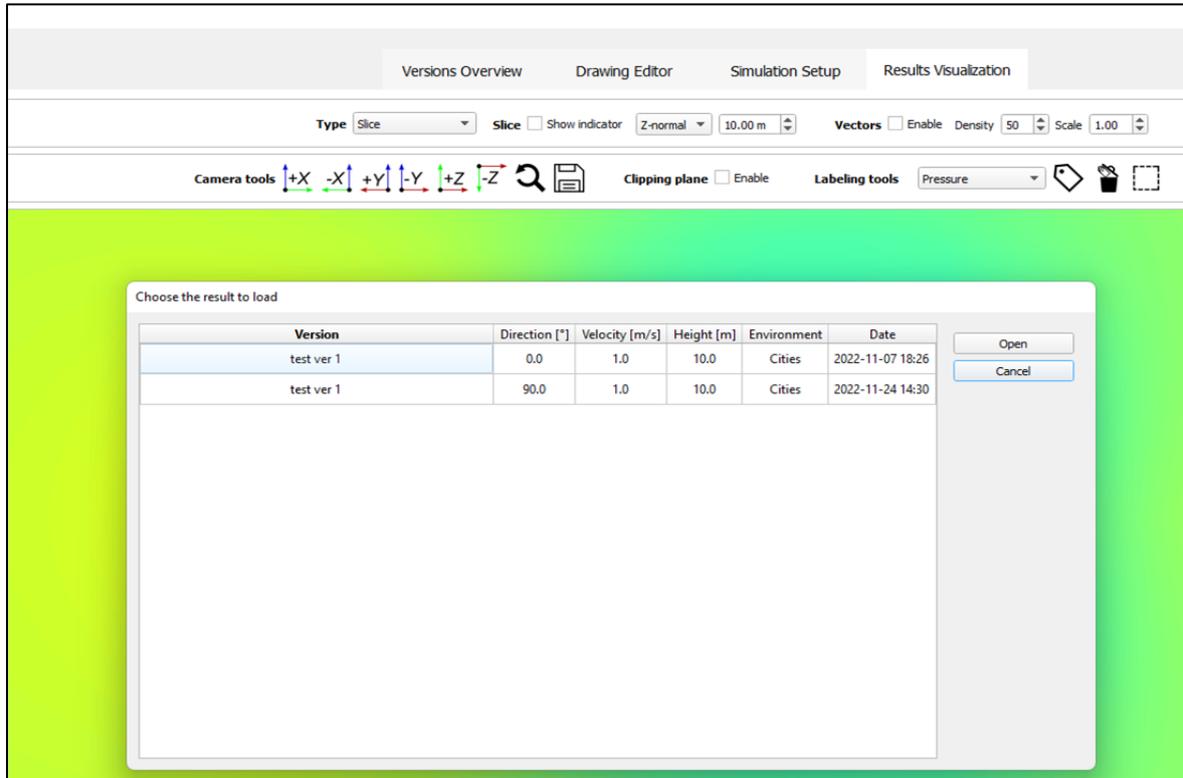


Figure 2-68 Choosing another simulation result to load

2.4.8 Visualizing the airflow through an opening

A new feature of this version of Vayu Pravah, allows the user to measure the volumetric airflow through an opening like a window or a shaft in the building. The user can click on the " " icon, place it on the opening in the simulation, and see the airflow.

In results visualization lowermost toolbar, select the icon with a left click.

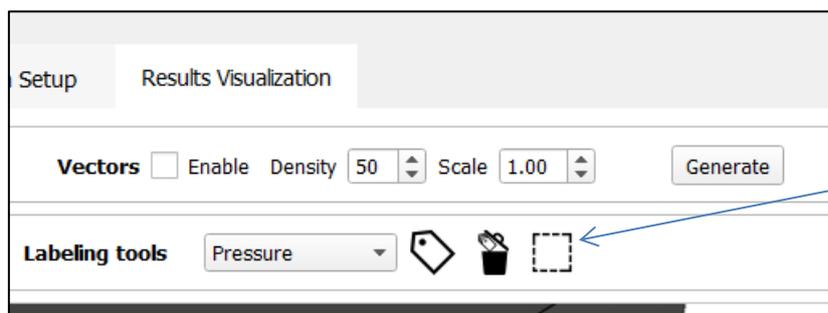


Figure 2-69 Visualizing airflow through an opening

In the results, with consequent left clicks, specify the points of a closed surface around the opening. (The closed surface should be larger than the opening) . As the surface is drawn, the selected area will be highlighted.

Then, right click to confirm the surface and see the flow rate magnitude and direction through the opening.

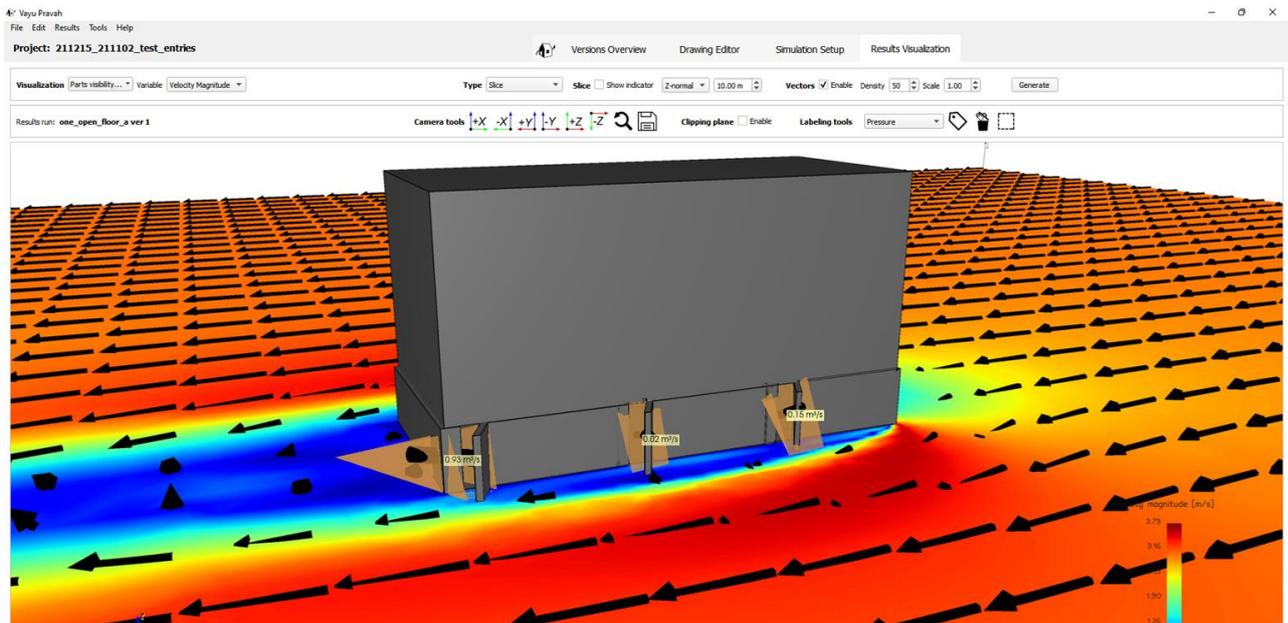


Figure 2-70 Analyzing airflow through openings in a building

2.4.9 Generate pdfs reports

The program allows the user 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 user can also see the preview of the pdf report in this window. The user can also click and drag to select the bounds(region) of the report.

In the pdf report, the user can also choose to add velocity vector arrows.

The pdfs can be used to systematically compare versions after calculations.

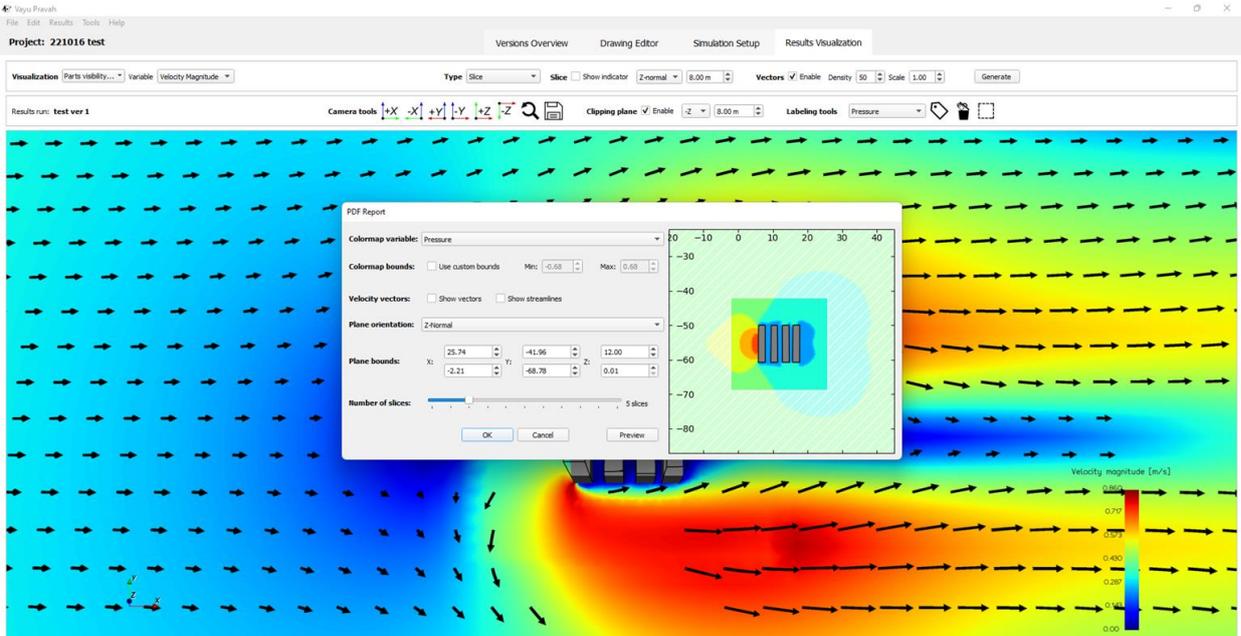


Figure 2-71 Preparing for a pdf report

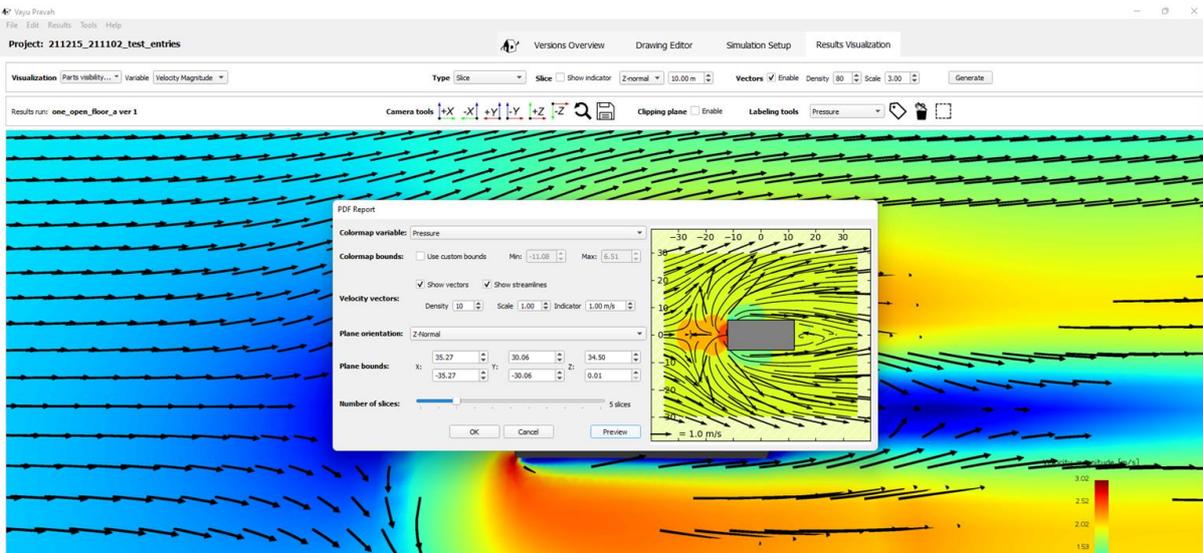


Figure 2-72 PDF report with vectors enabled

Select the location of the report to be saved.

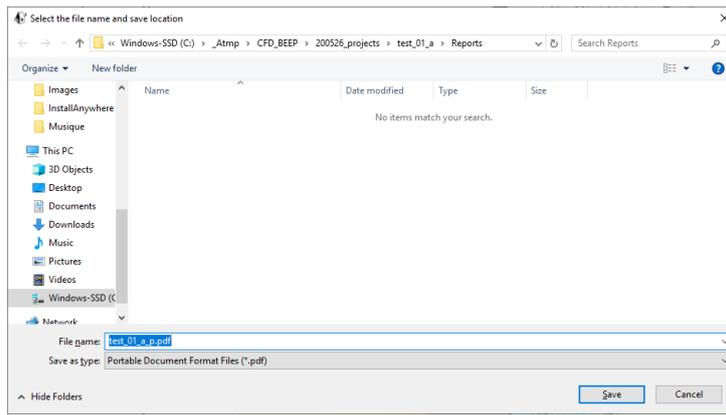


Figure 2-73 Saving the pdf report

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

3 Example with buildings

Let us now consider an example of a building in a project and use Vayu Pravah simulations to understand the airflow in the building.

Consider an example of a multi-storeyed building. To construct this building, we first import the different elements like the roof, the walls, the base and put them at suitable height from the ground.

First we enter the floor

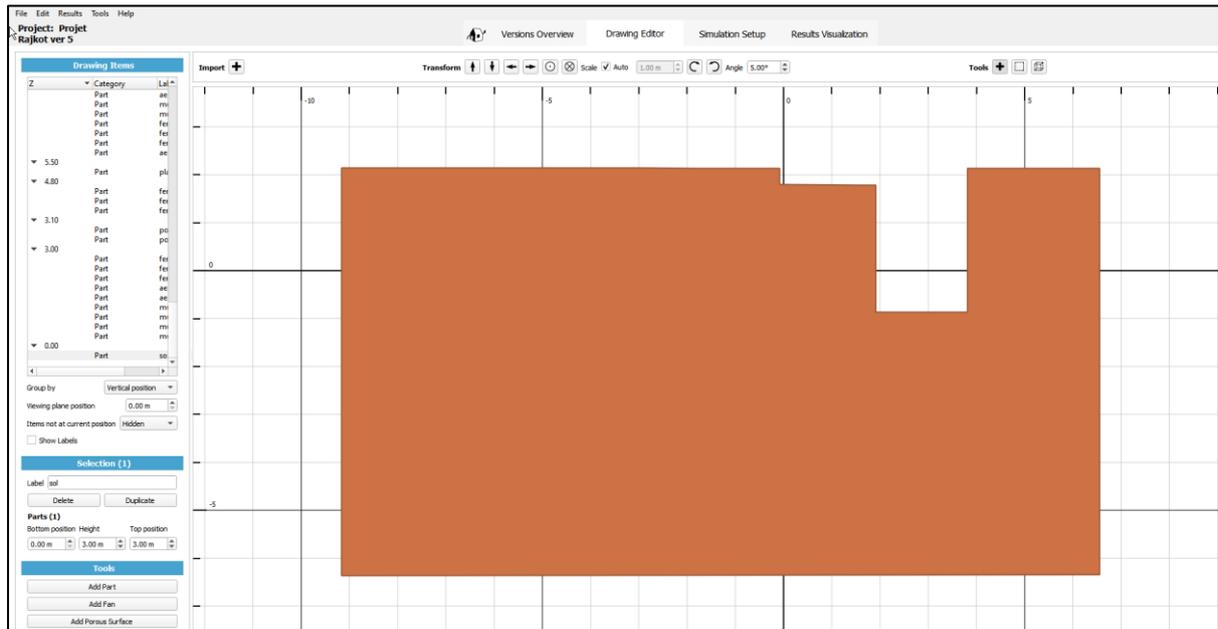
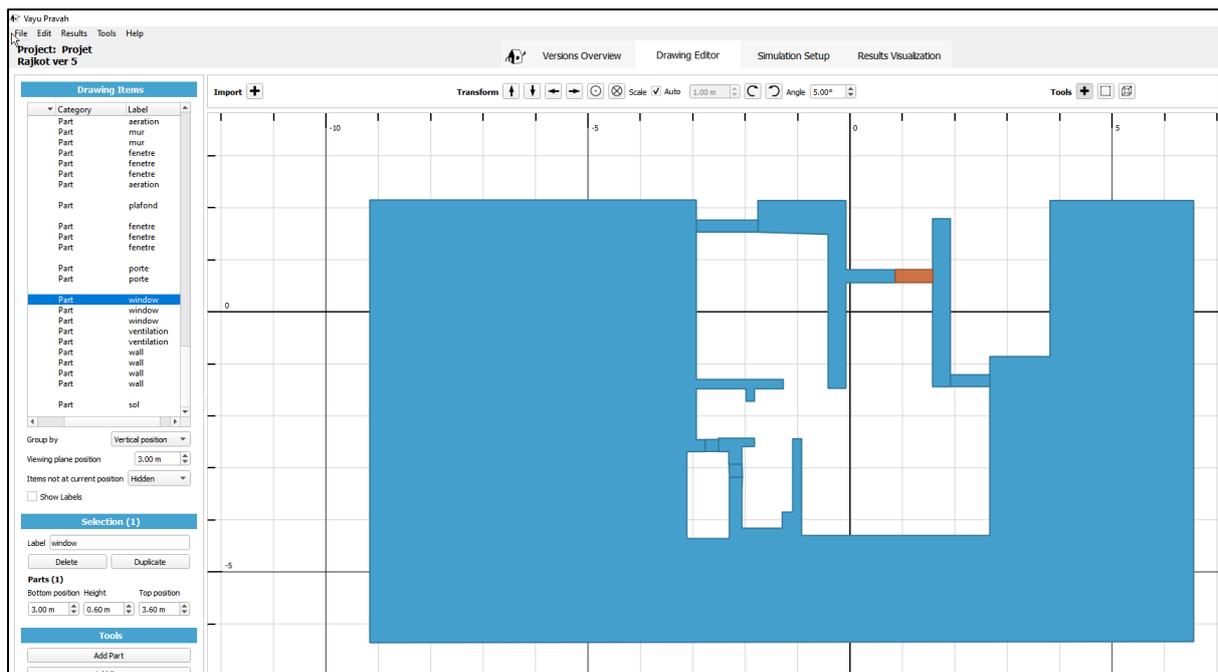
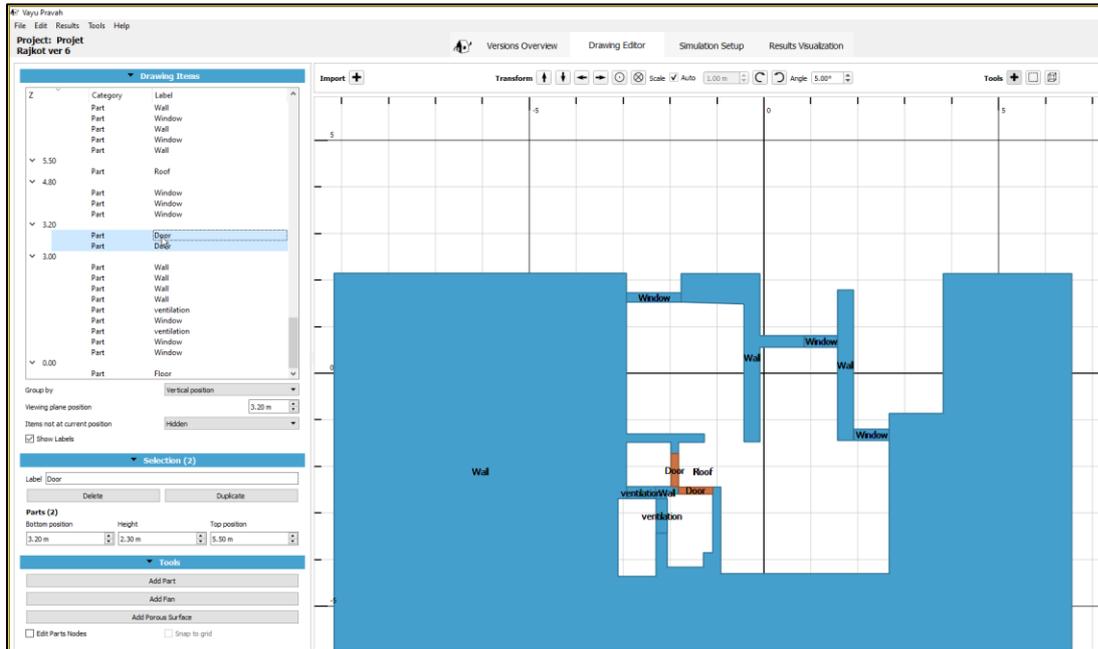


Figure 3-1 Enter the floor at vertical position 0

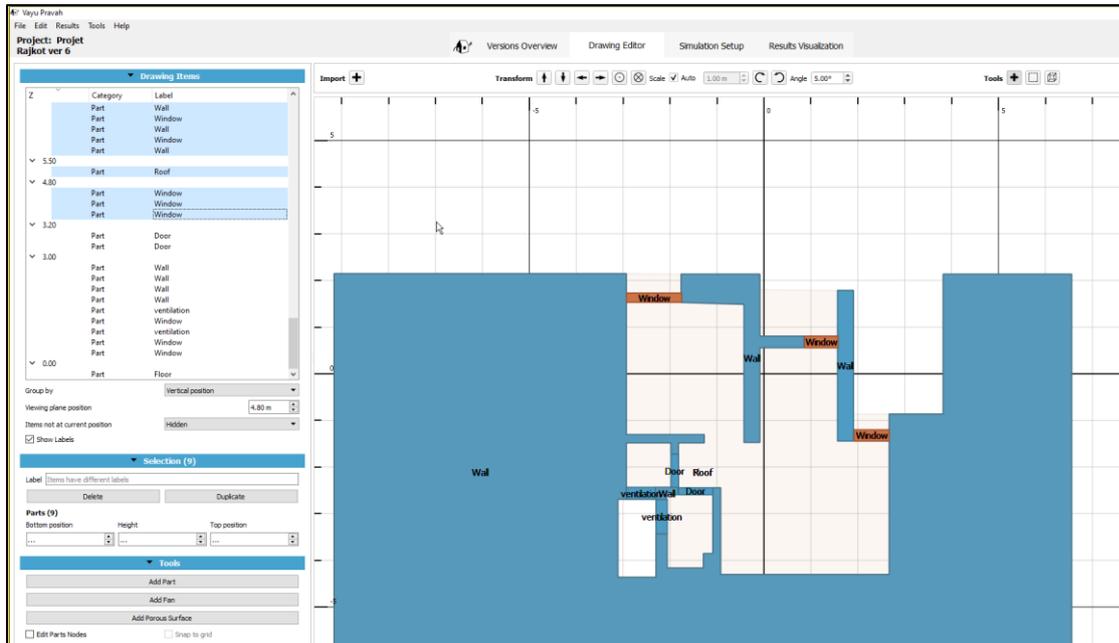
Then, at a height of 3 m, walls and windows are added along with the ventilation.



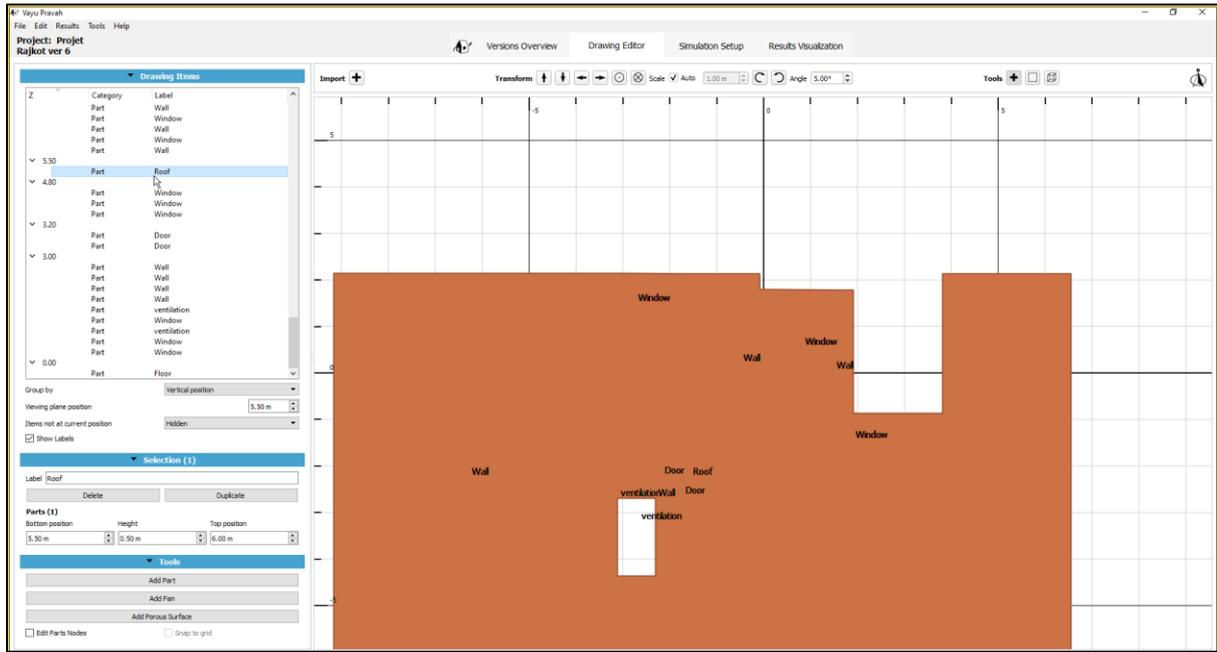
Two doors are added at 3.2 m



Windows are added at 4.8 m



Finally, the roof is added at 5m.



This makes the first floor of the building. Similarly other storeys can be added.

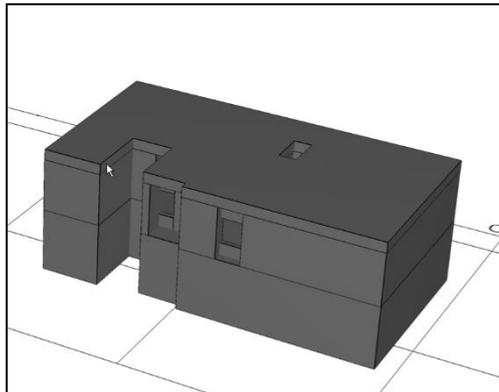


Figure 3-2 One floor of the building

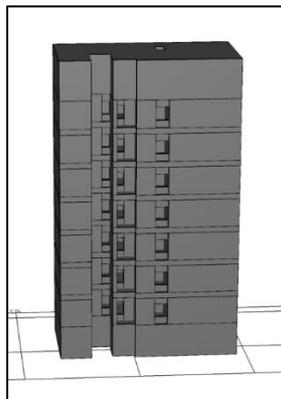


Figure 3-3 Entire building

A fan is added on the building top for forced ventilation and a simulation is run with the following parameters.

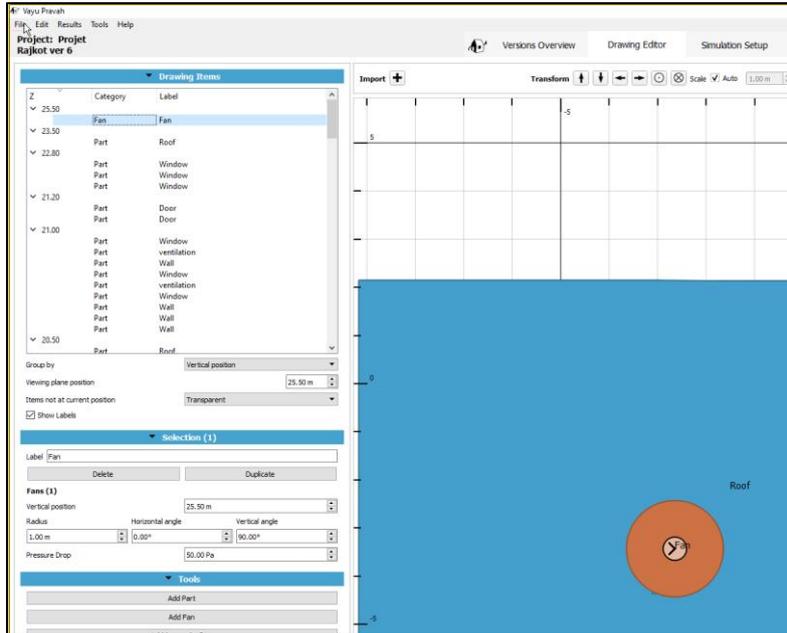


Figure 3-4 Adding a fan with its parameters

When simulation is run, the resulting airflow through the building can be visualized.

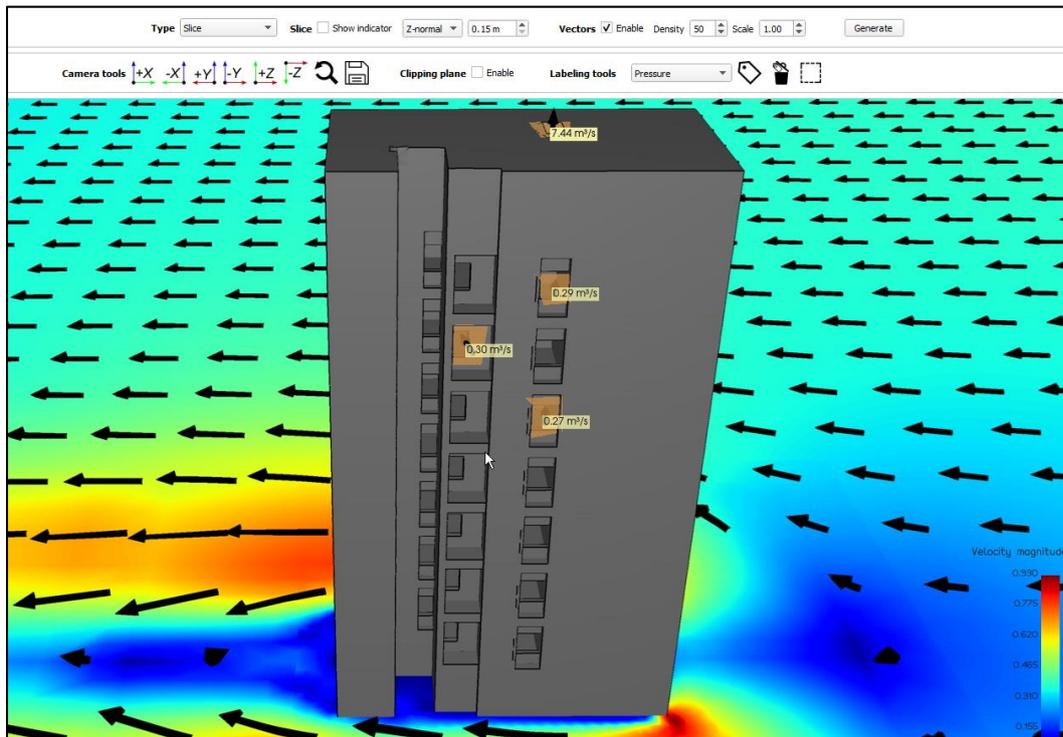


Figure 3-5 Airflow through the building with the added ventilation

4 Adding resistance

4.1 Simple equation for calculation of air across a window

4.1.1 Bernoulli Equation a refresher for an opened window without resistance

The Bernoulli's equation formula gives the relation between pressure, kinetic energy, and gravitational potential energy of a fluid.

Mathematically, it can be expressed as

$$p_1 + \frac{1}{2} \rho u_1^2 = p_2 + \frac{1}{2} \rho u_2^2$$

If one admits that the pressure differential across the opening is from ambient to a negative pressure in the zone, then the velocity far away is close to 0 so, the equation modifies as

$$p_1 = p_2 + \frac{1}{2} \rho u^2$$

so the velocity through the opening is equal to $u = \sqrt{2 \times \Delta P / \rho}$

by multiplying the velocity by the area of air passage, then one introduces a discharge coefficient (C_d) to take care of the real conditions of the air flow pattern (restriction, edge). Generally, for an open window without any resistance the value of $C_d=0.7$ is used.

$$Q = C_d * A * u = C_d * A * \sqrt{2 \times \Delta P / \rho}$$

$$C_d = 0.7$$

$$A = 2 \text{ m}^2$$

$$\Delta P = 5 \text{ Pa}$$

$$\rho = 1.2 \text{ kg/m}^3$$

$$Q = C_d * A * \sqrt{2 * \Delta P / \rho} \text{ m}^3/\text{sec}$$

$$Q = 4.04 \text{ m}^3/\text{sec} \text{ for a window opened without resistance (simplified Bernoulli equation)}$$

Then one does a simulation with a fan (see in corresponding paragraph in the manual) providing a negative pressure of 5 Pa in the zone and pulling the air through the opened window.

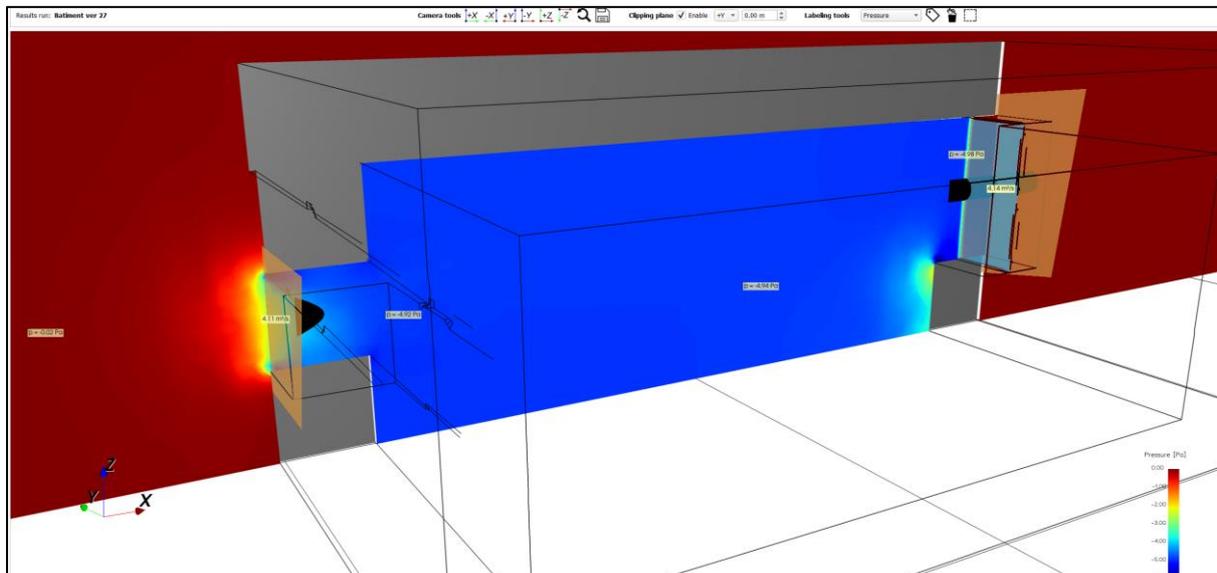


Figure 4-1 : the case calculated for a 2 m² window opened without additional resistance is shown above (air flow rate 4.12 m³/sec)

The result obtained compares well with the simplified Bernoulli equation.

4.1.2 Example with resistance and fan

Here we take the example of a building and add resistance of different porosity levels to it to show the effect of the resistance in the final airflow through the building.

Consider a building with a resistance (net), and a fan on top for forced ventilation. The resistance is added on the building at height of 10 m with porosity of 1 .

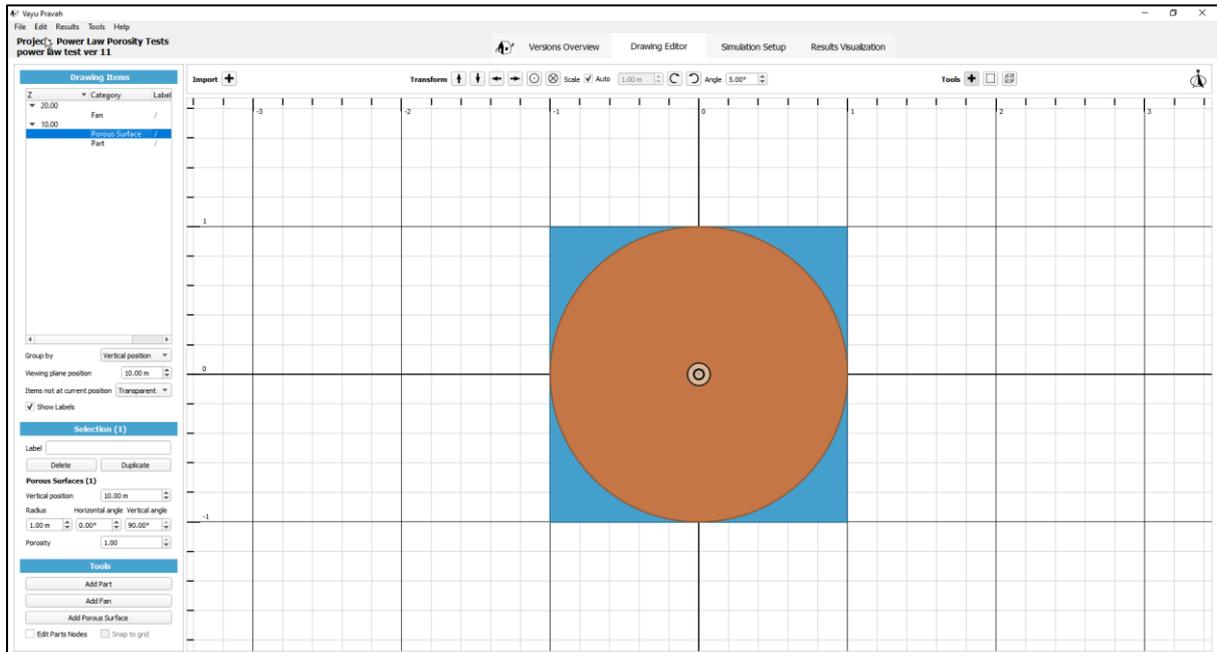


Figure 4-2 A porous surface is added at 10 m, and a fan at 20 m for forced ventilation.

The parameters defined for the porous surface and the fan are as follows.

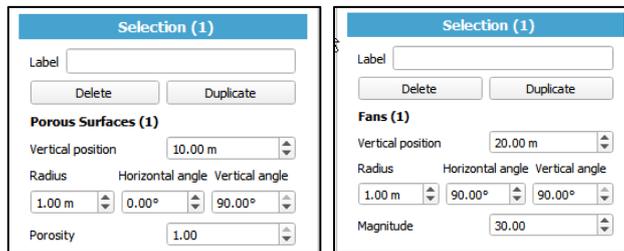
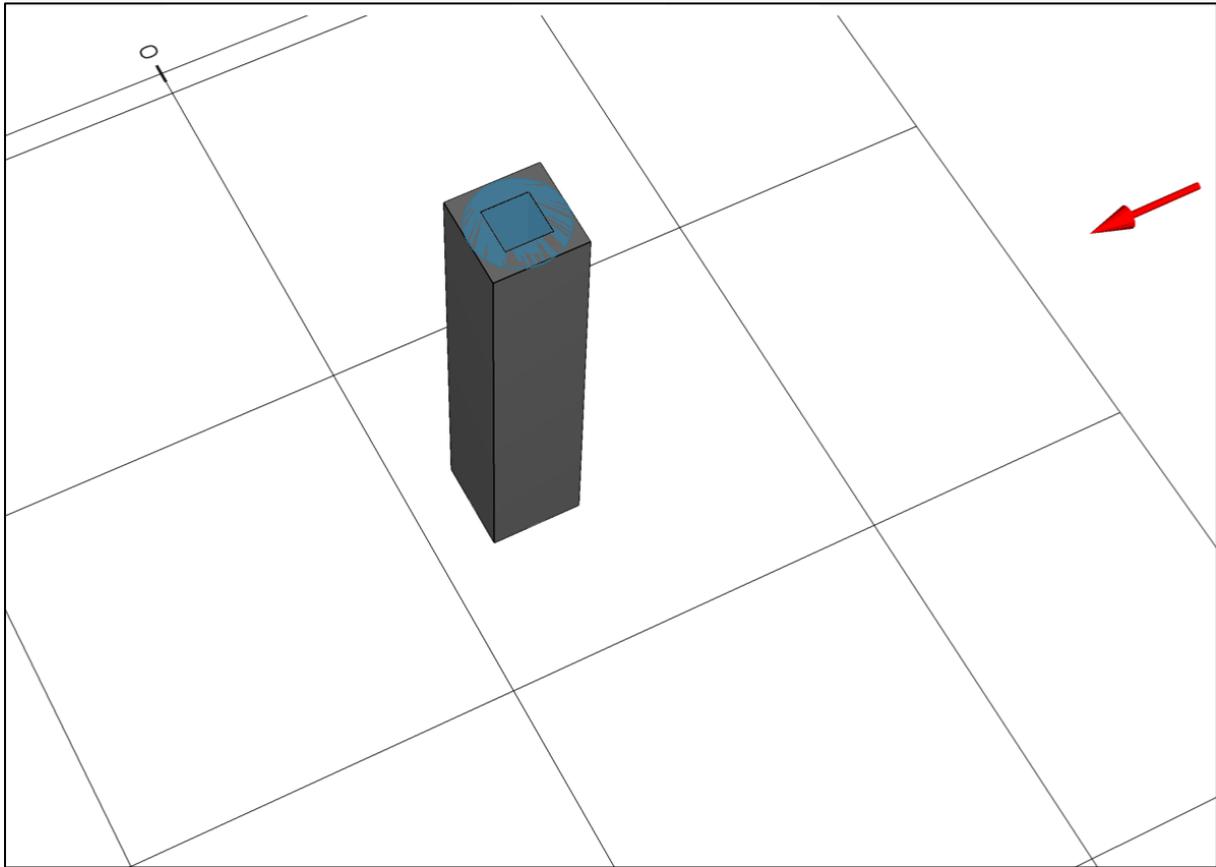


Figure 4-3 Parameters for the porous surface (on the left) and the fan

Then the simulation parameters are defined, and the simulation is run.



Drawing Items

Z	Category	Label
20.00	Fan	/
10.00	Porous Surface	/
	Part	/

Group by: Vertical position

Viewing plane position: 10.00 m

Items not at current position: Transparent

Show Labels

Selection (1)

Label:

Delete Duplicate

Porous Surfaces (1)

Vertical position: 10.00 m

Radius: 1.00 m Horizontal angle: 0.00° Vertical angle: 90.00°

Porosity: 1.00

Figure 4-4 Adding parameters of the simulation

After running the simulation, a top view of the velocity magnitude visualization is taken and the airflow is measured around the top. By changing the resistance value, the difference in this airflow can be seen for different cases.

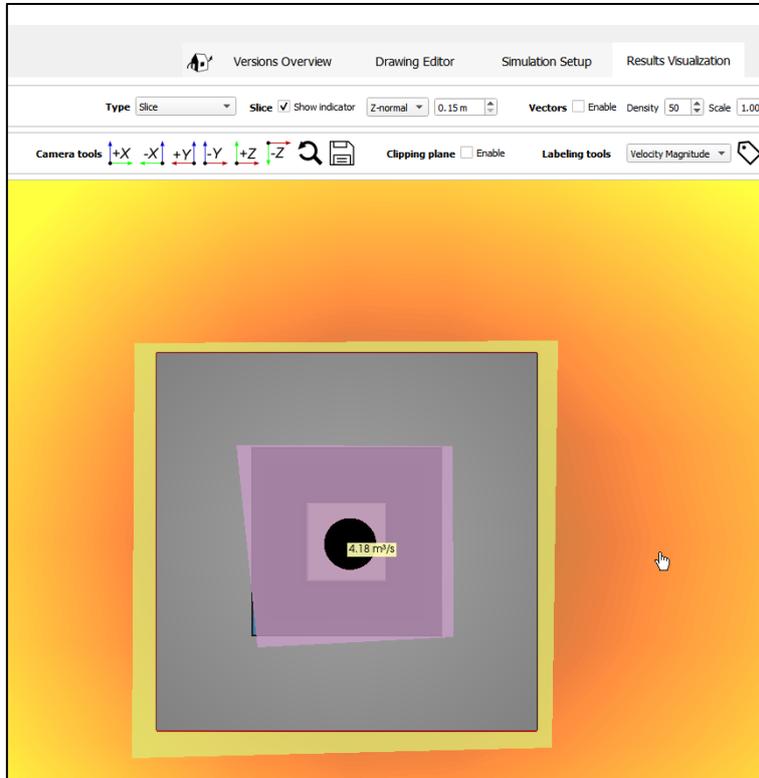
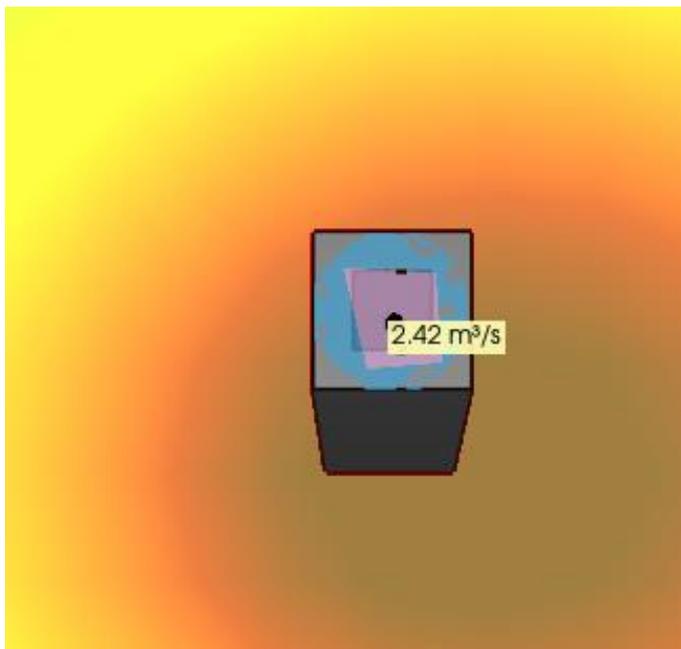


Figure 4-5 Airflow from the opening is visualized



4.1.3 Example with a resistance like mosquito net

First to do the simulation with an additional resistance, add a porous element, and select its resistance in the multiple-choice menu (with a resistance)

▼ Drawing Items

Z	Category	Label
3.00	Part	/
2.00	Fan	/
	Part	/
	Part	/
1.50	Porous Surface	porous 1
1.00	Part	/
	Part	/
0.00	Part	/

Group by: Vertical position
Viewing plane position: 1.50 m
Items not at current position: Transparent
 Show Labels

▼ Selection (1)

Label: porous 1
Delete Duplicate

Porous Surfaces (1)

Vertical position: 1.50 m
Radius: 1.50 m
Horizontal angle: 0.00°
Vertical angle: 0.00°
Porosity type: **Mosquito net (Cd ~ 0.43)**

▼ **T**
Add
Add Fan
Add Porous Surface

Edit Parts Nodes Snap to grid

Figure 4-6 : adding and selecting the porosity type of the porous element

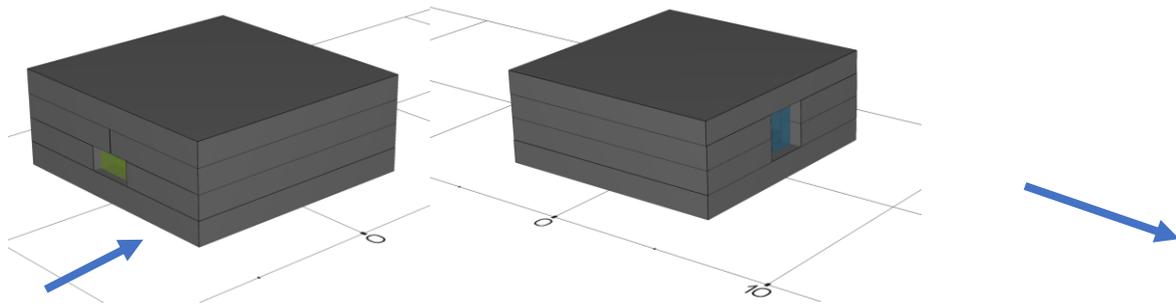


Figure 4-7 window opening with air entering and exhaust with a 5 Pa generating fan

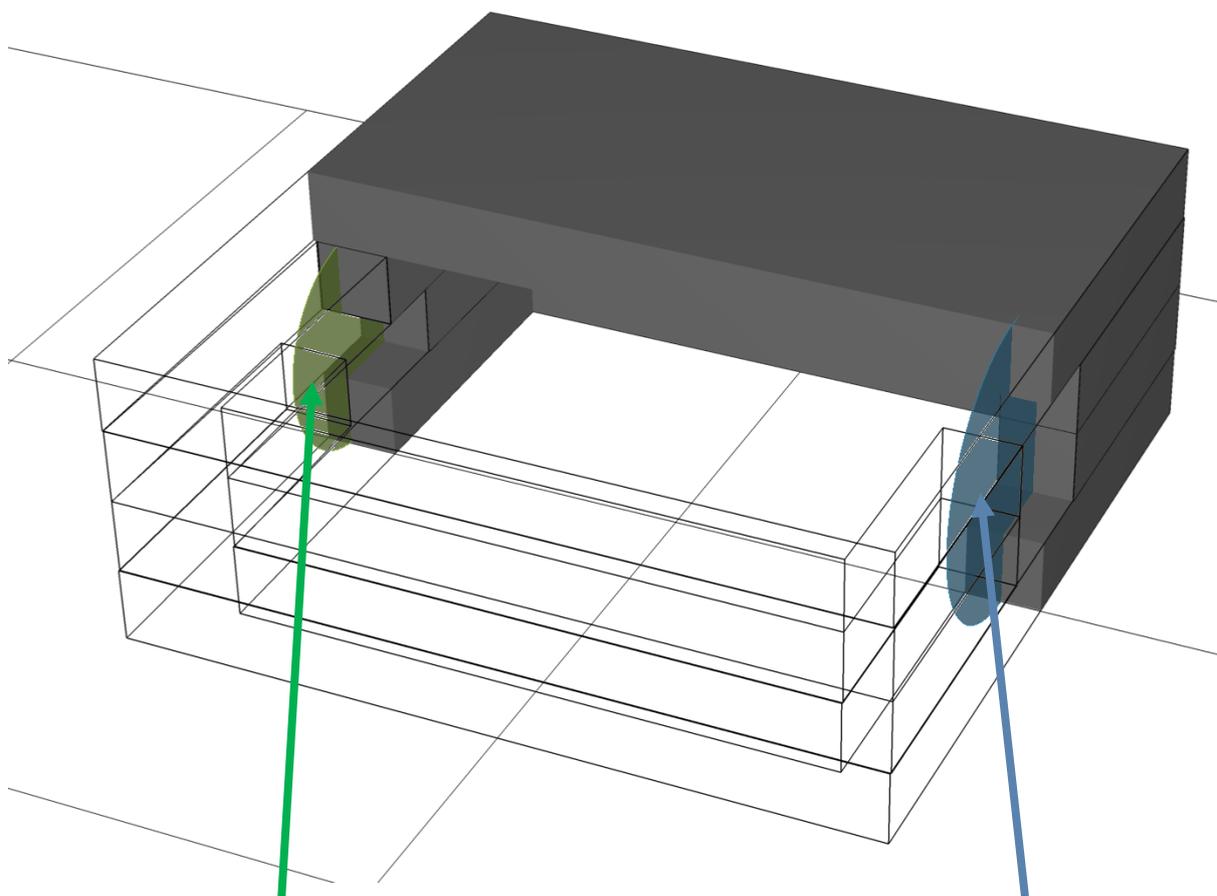


Figure 4-8 : view of the porous surface fan and of the

The opened area where the fan is applied is larger so that the velocity is small and so one gets a negative pressure very near to the -5 Pa desired.

Project: Porosity 2023 02 13
Batiment ver 30

Versions Overview Drawing Editor Simulation Setup Results Visualization

Import + Transform Scale Auto 1.00 m Angle 5.00° Tools

Drawing Items

Z	Category	Label
3.00	Part	/
2.00	Fan	/
	Part	/
	Part	/
1.50	Porous Surface	poro
1.00	Part	/
	Part	/
0.00	Part	/

Group by: Vertical position
Viewing plane position: 2.00 m
Items not at current position: Transparent
 Show Labels

Selection (1)

Label:
Delete Duplicate

Fans (1)

Vertical position: 2.00 m
Radius: 2.00 m
Horizontal angle: 0.00°
Vertical angle: 0.00°
Pressure Drop: 5.00 Pa

Tools

Add Part
Add Fan
Add Porous Surface

Hide Partic Nodes Show to draft

Project: Porosity 2023 02 13
Batiment ver 30

Versions Overview Drawing Editor Simulation Setup Results Visualization

Import + Transform Scale Auto 1.00 m Angle 5.00° Tools

Drawing Items

Z	Category	Label
3.00	Part	/
2.00	Fan	/
	Part	/
	Part	/
1.50	Porous Surface	porou
1.00	Part	/
	Part	/
0.00	Part	/

Group by: Vertical position
Viewing plane position: 1.50 m
Items not at current position: Transparent
 Show Labels

Selection (1)

Label: porous 1
Delete Duplicate

Porous Surfaces (1)

Vertical position: 1.50 m
Radius: 1.50 m
Horizontal angle: 0.00°
Vertical angle: 0.00°
Porosity type: Mosquito net (Cd = 0.43)

Tools

Add Part
Add Fan
Add Porous Surface

Hide Partic Nodes Show to draft

1.50	Porous Surface	porou
1.00	Part	/
	Part	/
0.00	Part	/

Group by: Vertical position

Viewing plane position: 1.50 m

Items not at current position: Transparent

Show Labels

Selection (1)

Label: porous 1

Delete Duplicate

Porosity Surfaces (1)

Vertical position: 1.50 m

Radius: 1.50 m Horizontal angle: 0.00° Vertical angle: 0.00°

Porosity type: Mosquito net (Cd ~ 0.43)

