

How to Use the CFD Software: PHOENICS (v 3.5)

Zhiqiang (John) Zhai Helen Xing Les Norford

PHOENICS is a general-purpose software package that predicts quantitatively how fluids (air, water, steam, oil, blood, etc) flow in and around, the associated changes of chemical and physical composition and the associated stresses in the immersed solids. In architecture, PHOENICS is used to predict the airflow in and around buildings, to improve the architectural design and thermal comfort. More information about PHOENICS can be found at <http://www.cham.co.uk>.

This little manual consists of two parts. General guidelines for using PHOENICS are introduced in the first part, general instructions. In the second part, case studies, we walk you through three examples: an outdoor airflow simulation case, an indoor simulation case, and an indoor simulation case with heat exchange.

PHOENICS is available on CD, to be copied into your personal computer. It is not currently available on departmental computers. If you choose to use the CD, please borrow it from the instructor or teaching assistant, install the program, and return the CD promptly. You will need to enter an access code to run it. Go to the PHOENICS directory, then subdirectory D_allpro, then open the config file using Notepad or Word, then change the software key to:

```
ID=:9STPSZP0+PXZG8A0TL8(/PSF=META1A
```

This software key is good for two months, as of December 4, 2002.

Note for Version 3.5.

This manual was prepared for Version 3.1. Version 3.5 comes with extensive and excellent documentation, including numerous test cases and step-by-step instructions. The examples in this manual have not been checked with Version 3.5 but are supplied as-is because they lay out an orderly way of simulating airflows around and through buildings, without and then with heat flows.

General Instructions:

1. Begin with a new case

File ⇒ New case ⇒ VR Case ⇒ Core
Press 'OK'

2. Edit/Create your case in VR Editor

(1) Main Menu

Click on Main Menu and set up your modeling environment as follows:

Models ⇒ Turn 'ON' Solutions for velocity and pressure;
Select Turbulence Models (KERNG is a commonly used model);
Pickup 'TEMPERATURE' and 'TOTAL' if the simulation involves energy and temperature

Numerics ⇒ Choose '20' as your Total number of iterations for the first try.
(Usually it will take at least 200 steps to reach an accurate result).

Geometry ⇒ Define your Domain Size
Top menu ⇒ OK

(2) New Object

Click on New Object and start building your model

Name: give your object a name, such as building, inlet, etc.

X/Y/ZPos, X/Y/ZSize: set your object position and size

Type: choose your object type (inlet for your incoming wind; outlet for outgoing wind; blockage for buildings or other object).

Attribute: when selecting 'inlet' type, you need to set the incoming velocity at the opening. Through this window, you can also import a CAD file as your new object.

CAD Interface ⇒ Import geometry from an STL file ⇒ select your STL file.

Note 1: The software FormZ can generate a STL file, or convert your 'DWG' file to a STL file successfully. When you save your STL file, you need to choose the file type as 'ASCII'.

Note 2: After you import your CAD object, a black screen will show the size of your CAD object. You need to set this size as your object size.

Note 3: It is strongly recommended for beginners to build simple models in PHOENICS, run them and get some sense how the software works. Most models created in AutoCAD, looking simple in architects' eyes, are still quite complex for PHOENICS. Running those models in PHOENICS takes a lot of computation time (could be days) and experience to get convergence.

(3) *Main Menu*

Once objects have been created, come back to the main menu and mesh the computation region.

Geometry ⇒ Modify region ⇒ Cell: set the cell number ⇒ OK

Note: Grid dimension is your total cell number along one direction. You should limit your total cell number (the product of the grid dimensions along X, Y, and Z directions) to be less than 200,000. It is suggested to set grid dimensions in the X, Y and Z directions proportionally to their relative size. The larger the cell number, the more detailed information you will know about the flow condition in that region but the longer the computation will take.

(4) *Exit* ⇒ “Yes” to exit VR editor ⇒ “Yes” to save your data

3. Run your case in Earth

Once you start a simulation, there will be two windows on your screen. The one on the left contains the values of the monitoring point and the lines are expected to be flat toward the end of the simulation, indicating convergence. The one on the right shows the simulation errors at the monitoring point and the lines are expected to go down during the simulation. These trends indicate that the simulation results are stable, which is exactly what we want.

4. View your results in VR Viewer

Choose ‘V’ (below Variables) to view your velocity field. The red button and the arrow button represent the different ways to show the results. The arrow is important. The buttons labeled X, Y and Z slice the model in the X, Y and Z directions, respectively, and allow you to view the velocity/temperature distribution on a specified plane.

5. Save case

File ⇒ Save case As ⇒ New Project:
New Case:

Note1: Although you can view your result, PHOENICS does not save your case automatically. You must ‘save case’ after you finish your simulation.

Note2: Your project will be saved in the folder of Phoenics/D_priv1/

Your input data is saved in ‘q1’ file

Your output data is saved in ‘phi’ file for visualization

5. Export image

PHOENICS has no image export function in version 3.1. You must use the “PrintScreen” button on the keyboard to copy an image, paste it in a new file created in a graphics software package (Photoshop or could be as simple as Paint in Accessories), and edit the image in the graphics software.

6. Open existing case

Change dir (must be in PHOENICS sub-directory)

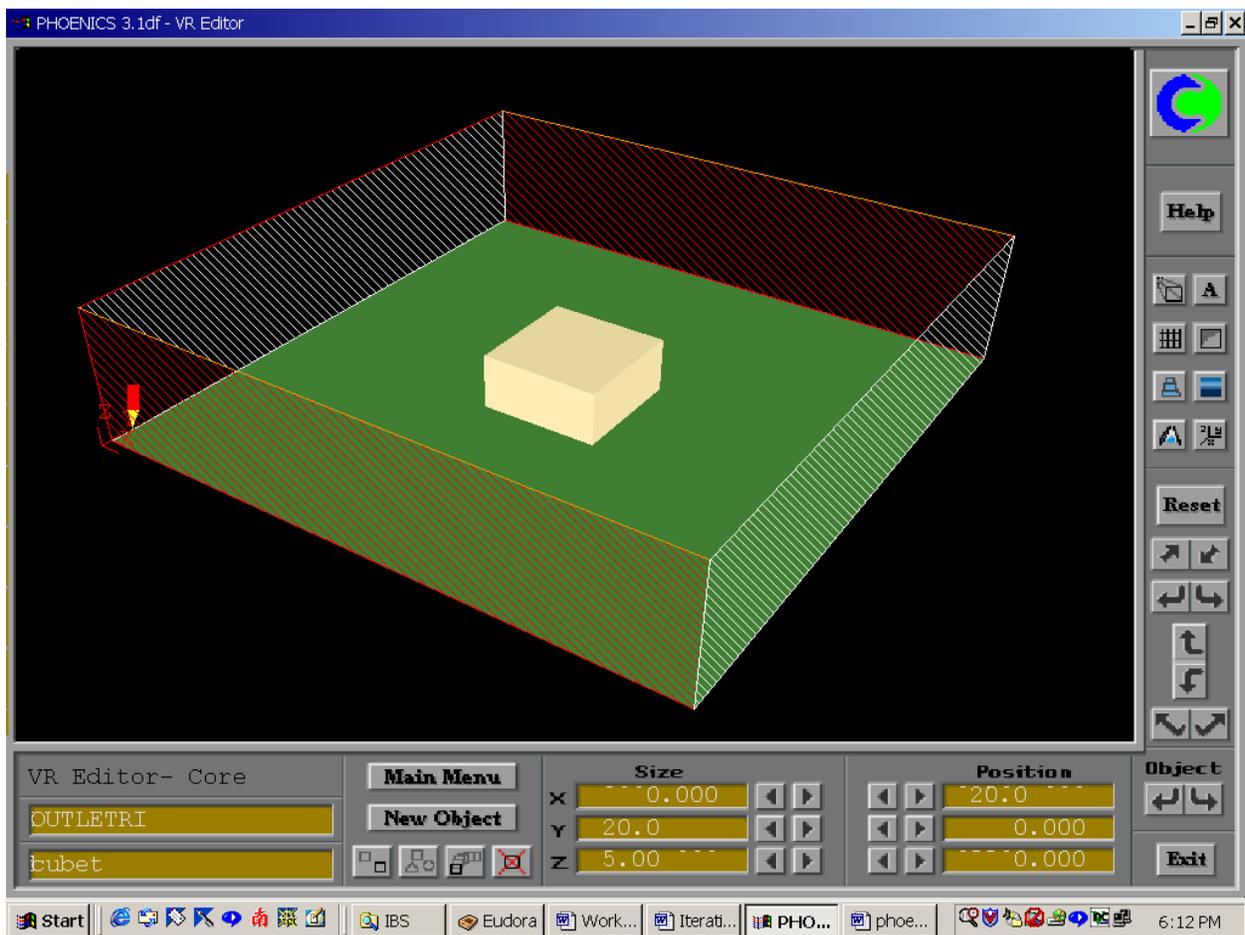
File ⇒ Open user’s existing case ⇒ Choose a project

Press ‘OK’

Step-By-Step Instructions on Demo Cases

1) *Outdoor case*

In this case, we study the airflow around a rectangular building (length x width x height = 4 x 4 x 2 meter). The computational domain size is 20 x 20 x 5 meter. The wind comes from the south (x-z plane) with the uniform velocity of 3 m/s. The other boundaries are natural atmosphere condition, except the ground as solid surface. The case is shown as follows.



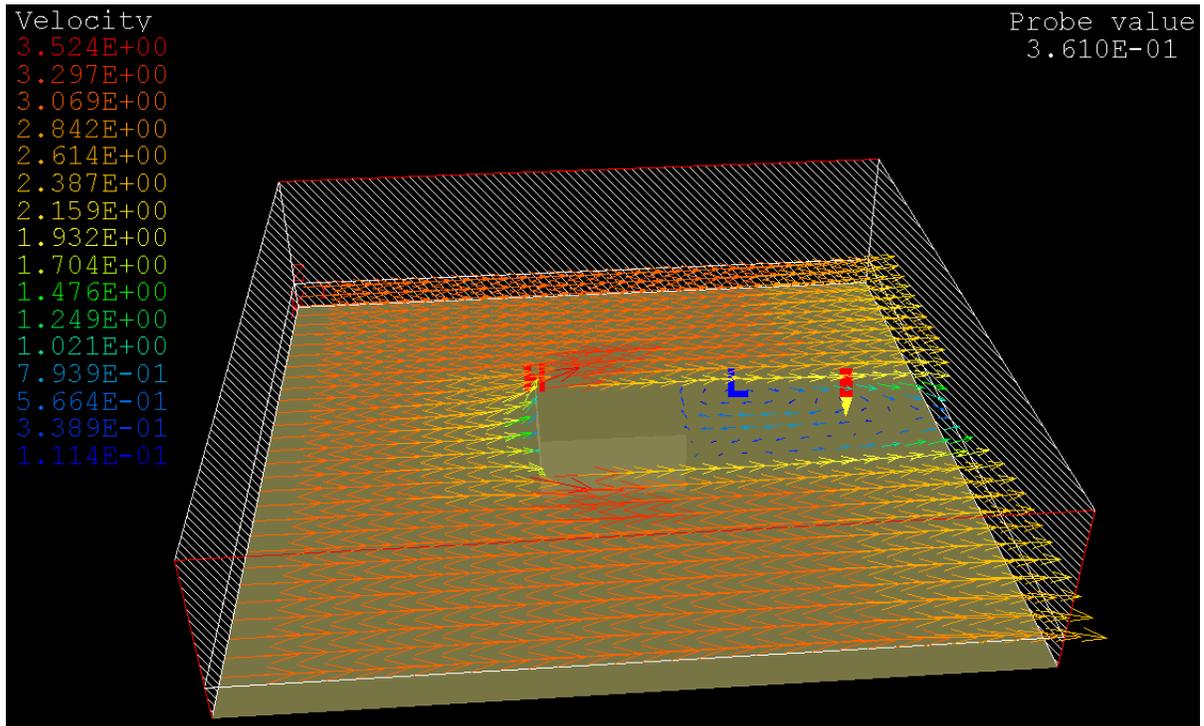
1. Menu: File \Rightarrow New case \Rightarrow VR Case \Rightarrow Core, Press “OK”;
2. Enter “VR Editor”;
3. Click “Main Menu” ;
 - a. Title of current simulation: Outdoor Demo
 - b. Geometry \Rightarrow Domain Size: X=20 (unit: m), Y=20, Z=5;
 - c. Models \Rightarrow Solution for velocities and pressure: ON; Turbulence Models: KERNG;
4. Click “Top menu” then “OK” to quit “Main Menu”;
5. Click “New Object” to set up the ground;
 - a. Name: Ground;

- b. Xposition: 0; Xsize: 20;
 - c. Yposition: 0; Ysize: 20;
 - d. Zposition: 0; Zsize: 0;
 - e. Geometry \Rightarrow Shapes \Rightarrow cube1 (Select different colors and patterns to better view objects. This is convenient, not essential.)
 - f. Type: PLATE;
 - g. "OK";
6. Click "New Object" to set up a building
 - a. Name: Building;
 - b. Xposition: 8; Xsize: 4;
 - c. Yposition: 8; Ysize: 4;
 - d. Zposition: 0; Zsize: 2;
 - e. Type: BLOCKAGE;
 - f. "OK";
 7. Click "New Object" to set up the sky;
 - a. Name: Sky;
 - b. Xposition: 0; Xsize: 20;
 - c. Yposition: 0; Ysize: 20;
 - d. Zposition: 5; Zsize: 0;
 - e. Geometry \Rightarrow Shapes \Rightarrow cubet (select different colors for better view)
 - f. Type: OUTLET;
 - g. "OK";
 8. Click "New Object" to set up the upwind inlet;
 - a. Name: Inlet;
 - b. Xposition: 0; Xsize: 20;
 - c. Yposition: 0; Ysize: 0;
 - d. Zposition: 0; Zsize: 5;
 - e. Geometry \Rightarrow Shapes \Rightarrow cubet1 (select different colors for better view)
 - f. Type: INLET;
 - g. Attributes \Rightarrow Y Direction: 3 m/s \Rightarrow "OK";
 - h. "OK";
 9. Click "New Object" to set up the downwind outlet;
 - a. Name: Outlet;
 - b. Xposition: 0; Xsize: 20;
 - c. Yposition: 20; Ysize: 0;
 - d. Zposition: 0; Zsize: 5;
 - e. Geometry \Rightarrow Shapes \Rightarrow cubet1 (select different colors for better view)
 - f. Type: OUTLET;
 - g. "OK";
 10. Click "New Object" to set up an outlet on the left side;
 - a. Name: OutletLeft;
 - b. Xposition: 0; Xsize: 0;
 - c. Yposition: 0; Ysize: 20;
 - d. Zposition: 0; Zsize: 5;
 - e. Geometry \Rightarrow Shapes \Rightarrow cubet1 (select different colors for better view)
 - f. Type: OUTLET;
 - g. "OK";

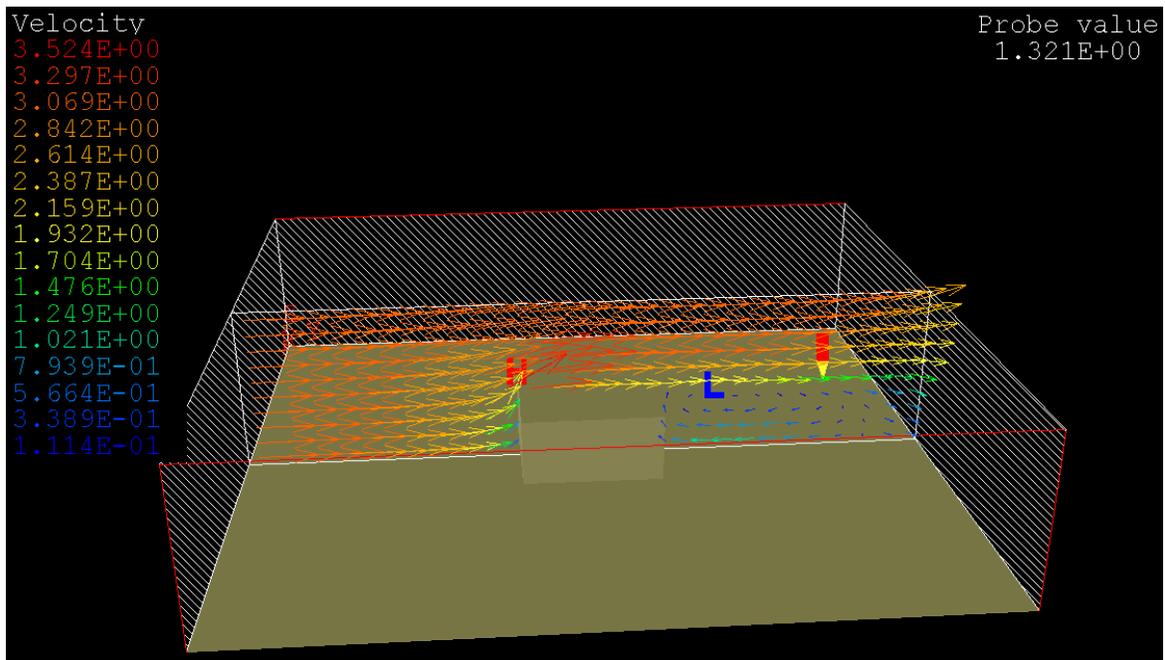
11. Click “New Object” to set up an outlet on the right side;
 - a. Name: OutletRight;
 - b. Xposition: 20; Xsize: 0;
 - c. Yposition: 0; Ysize: 20;
 - d. Zposition: 0; Zsize: 5;
 - e. Geometry \Rightarrow Shapes \Rightarrow cubet1 (select different colors for better view)
 - f. Type: OUTLET;
 - g. “OK”;
12. Enter “Main Menu” again;
 - a. Geometry \Rightarrow Grid Dimension: X=30, Y=30, Z=10;
 - b. Initialization \Rightarrow V1=3 m/s (since Vinlet=3m/s); KE=0.001; EP=0.0001;
 - c. Numerics \Rightarrow Total number of iterations: 100;
 - d. Numerics \Rightarrow Relaxation Control \Rightarrow P1: -0.5; U1, V1, W1, KE, EP: 1; \Rightarrow Previous panel;
 - e. Output \Rightarrow XMON: 10; YMON: 17; ZMON: 2.5 (unit: m) (monitor point of calculation. The bottom line is to make sure the monitoring point is in your simulation region);
13. Click “Top menu” then “OK” to quit “Main Menu”;
14. “Exit”;
15. “Do you want to exit VR Editor?” \Rightarrow “Yes”;
16. “Do you want to save date?” \Rightarrow “Yes”;
17. Click “Earth” to run ...
18. Once done, use “ VR Viewer” to view the results.

The velocity profiles for slices of $z = 1$ m and $x = 10$ m are shown below. The list in the left upper corner shows how the image colors and the numerical values are related. The number in the right upper corner is the probe value and the pencil indicates the location of the probe. The probe value and its sign change from case to case, depending on where the probe is located and whether the velocity at the probe is positive. H and L mean the highest and lowest values respectively.

Case 1: Velocity distribution on the horizontal plane $z = 1\text{ m}$

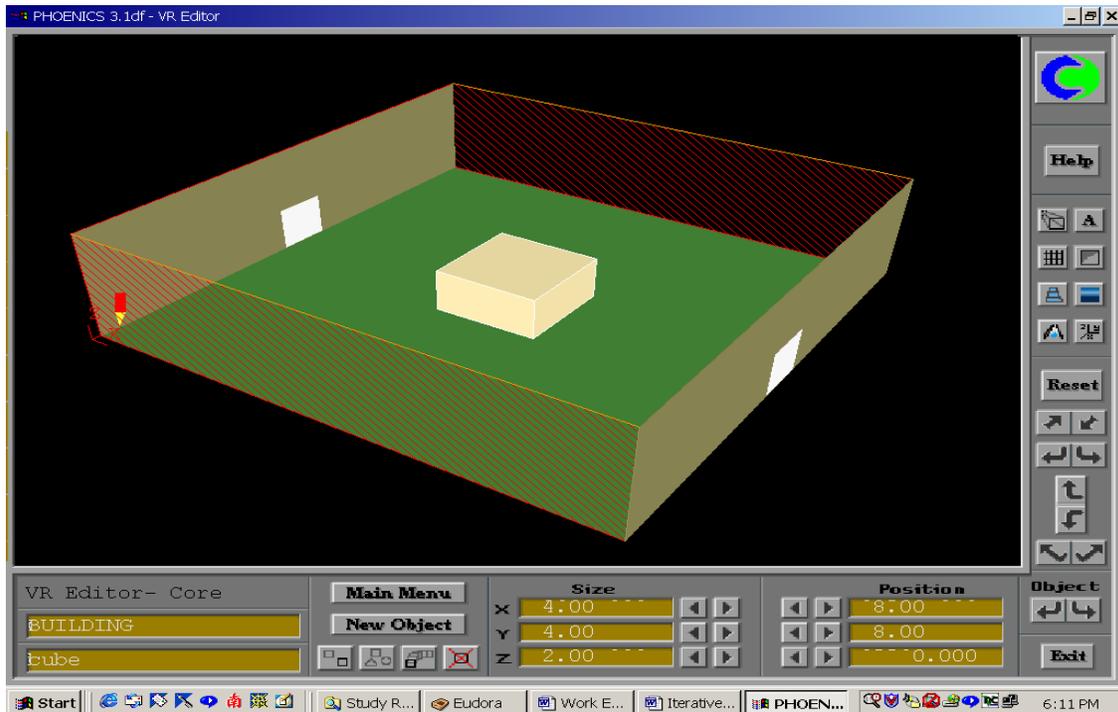


Case 1: Velocity distribution on the vertical plane $y = 10\text{ m}$



2) Indoor case without heat

We can modify case 1, which simulates the airflow around a building, for our indoor simulation purposes. The geometry and sizes remain the same. The difference is that we are now thinking of the whole thing as a room, say a machine shop with a huge machine (the building in case1) inside, and we are simulating the airflow indoors. The five free surfaces in case1 now need to be modified to form enclosures of the room. The indoor airflow is caused by the air-conditioning/ventilating system of a diffuser-exhaust pair, as shown below.



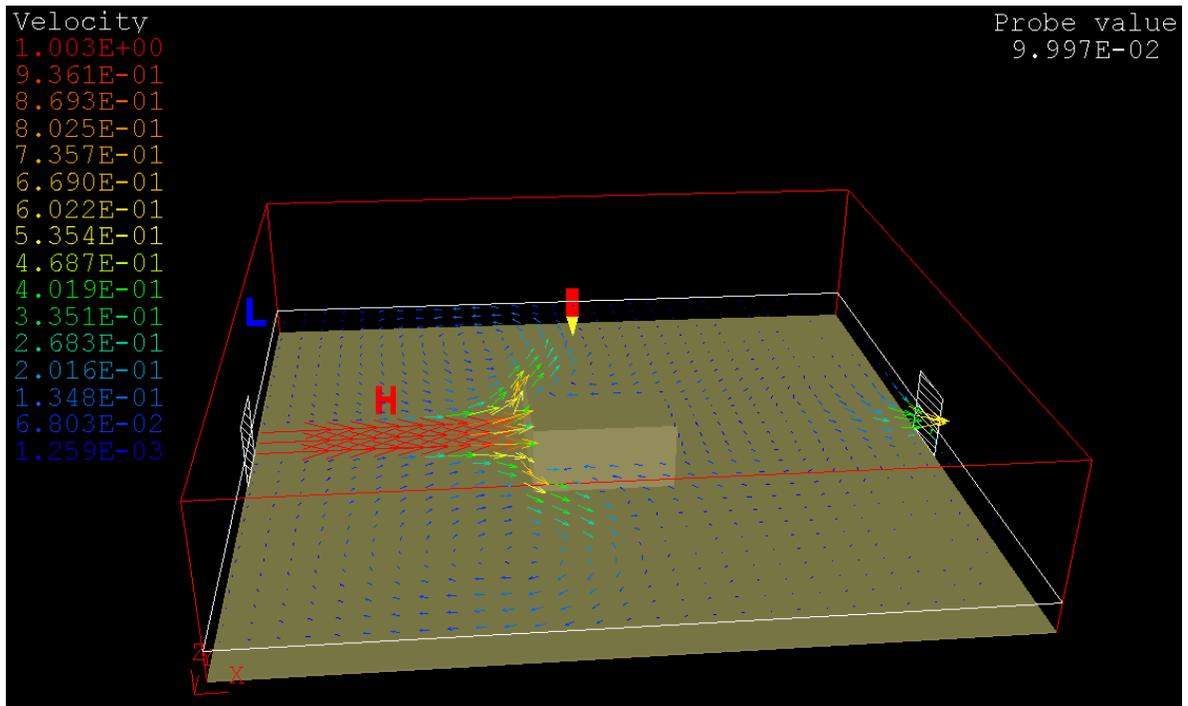
1. Enter "VR Editor";
2. Double click each surface to make them be walls;
 - Double click "Sky";
 - Name: Roof;
 - Type: PLATE;
 - "OK";

 - Double click "Inlet";
 - Name: South;
 - Type: PLATE;
 - "OK";

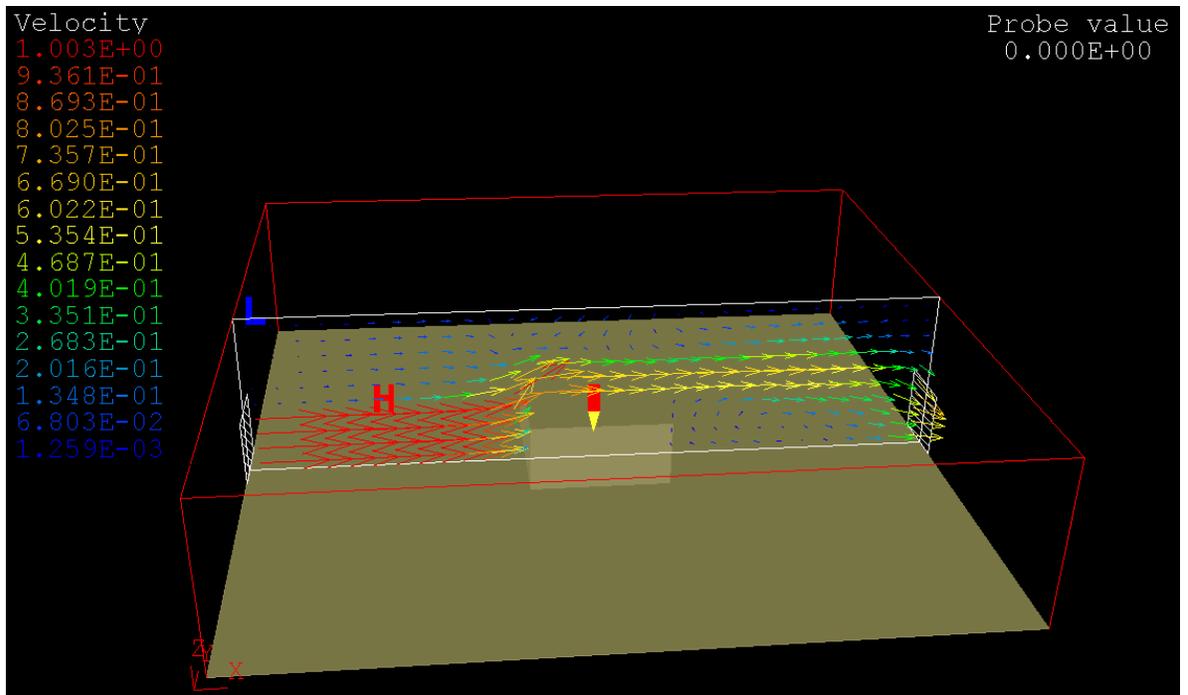
 - Double click "Outlet";
 - Name: North;
 - Type: PLATE;
 - "OK";

- Double click “OutletLeft”;
 - Name: West;
 - Type: PLATE;
 - “OK”;
- Double click “OutletRight”;
 - Name: East;
 - Type: PLATE;
 - “OK”;
3. Click “New Object” to setup Diffuser Inlet;
 - a. Name: Diffuser;
 - b. Xposition: 0; Xsize: 0;
 - c. Yposition: 9; Ysize: 2;
 - d. Zposition: 0; Zsize: 2;
 - e. Geometry \Rightarrow Shapes \Rightarrow cubet1 (select different colors for better view)
 - f. Type: INLET;
 - g. Attributes \Rightarrow X Direction: 1 m/s \Rightarrow “OK”;
 - h. “OK”;
 4. Click “New Object” to setup Exhaust Outlet;
 - a. Name: Exhaust;
 - b. Xposition: 20; Xsize: 0;
 - c. Yposition: 9; Ysize: 2;
 - d. Zposition: 0; Zsize: 2;
 - e. Geometry \Rightarrow Shapes \Rightarrow cubet1 (select different colors for better view)
 - f. Type: OUTLET;
 - g. “OK”;
 5. Enter “Main Menu” \Rightarrow Initialization \Rightarrow U1=1 m/s (since Uinlet=1m/s); KE=0.001; EP=0.0001;
 6. Same as Demo-1 (13) – (18).

Case 2: Velocity distribution on the horizontal plane $z = 1$



Case 2: Velocity distribution on the vertical plane $y = 10\text{m}$



3) *Indoor case with heat transfer*

This case is the extension of case 2. It involves heat transfer and the buoyancy effect. The enclosures are insulated, except that there is a hot roof (temperature assumed constant) due to the solar radiation. There is a 1500 W copier in the room. The diffuser supplies cold air at 15 °C to cool down the space.

1. Enter “VR Editor”;
2. Enter “Main Menu”;
 - a. Models ⇒ Energy Equation: TEMPERATURE TOTAL;
 - b. Sources ⇒ Gravitational forces: ON; Buoyancy model is: BOUSSINESQ;
Gravitational acceleration: X=0; Y=0, Z=-9.81;
 - c. Click “Top menu” then “OK” to quit “Main Menu”;
3. Double click each object to add heat info;
 - a. Double click “Roof”;
 - b. Attributes: temperature: 30 (unit: C); “OK”;
 - c. “OK”;

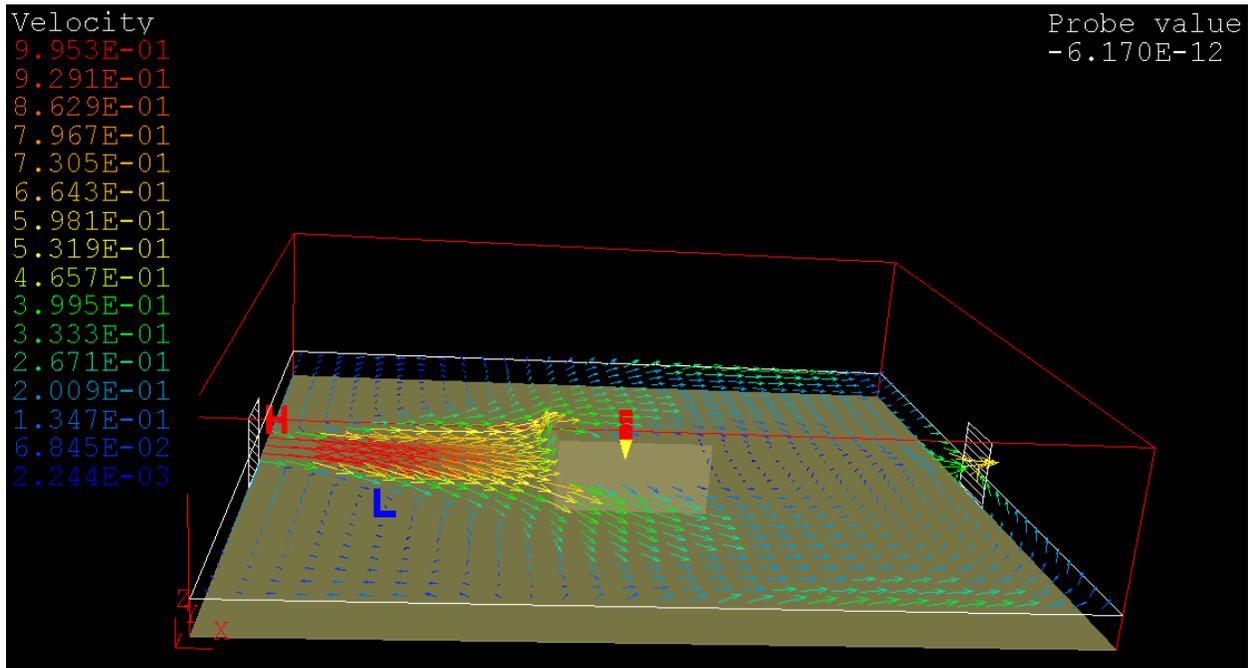
 - d. Double click “Ground”, “West”, “East”, “South”, “North”;
 - e. Attributes: heat source: 0 (unit: W) [insulated]; “OK”;
 - f. “OK”;

 - g. Double click “Diffuser”;
 - h. Attributes: temperature: 15 (unit: C); “OK”;
 - i. “OK”;

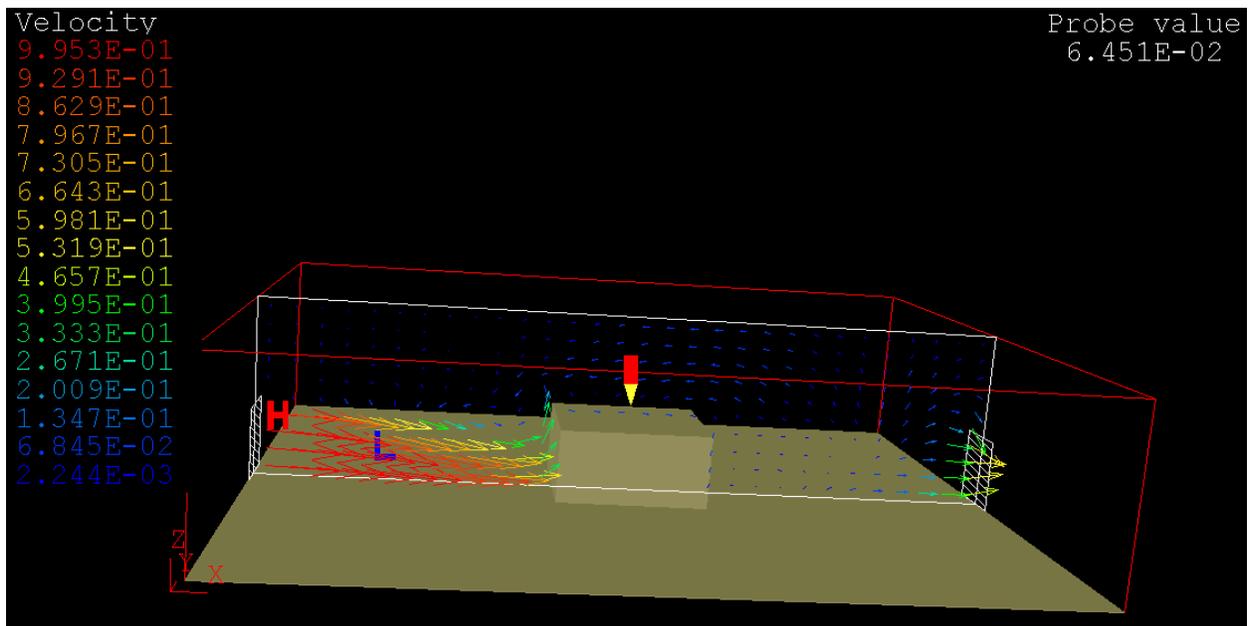
 - j. Double click “Exhaust”;
 - k. Attributes: temperature: 25 (if known, otherwise leave “0”); “OK”;
 - l. “OK”;

 - m. Double click “Building”;
 - n. Name: Copier
 - o. Attributes ⇒ Materials ⇒ Solids ⇒ 100 Aluminium ⇒ Heat Source: 1500 (unit: W); “OK”;
 - p. “OK”;
4. Enter “Main Menu” again;
 - a. Initialization ⇒ TEM1=20 (C) (guess, better than nothing);
 - b. Numerics ⇒ Relaxation Control ⇒ TEM1=1 ⇒ Previous panel;
 - c. Click “Top menu” then “OK” to quit “Main Menu”;
5. Same as Demo-1 (14) – (18).

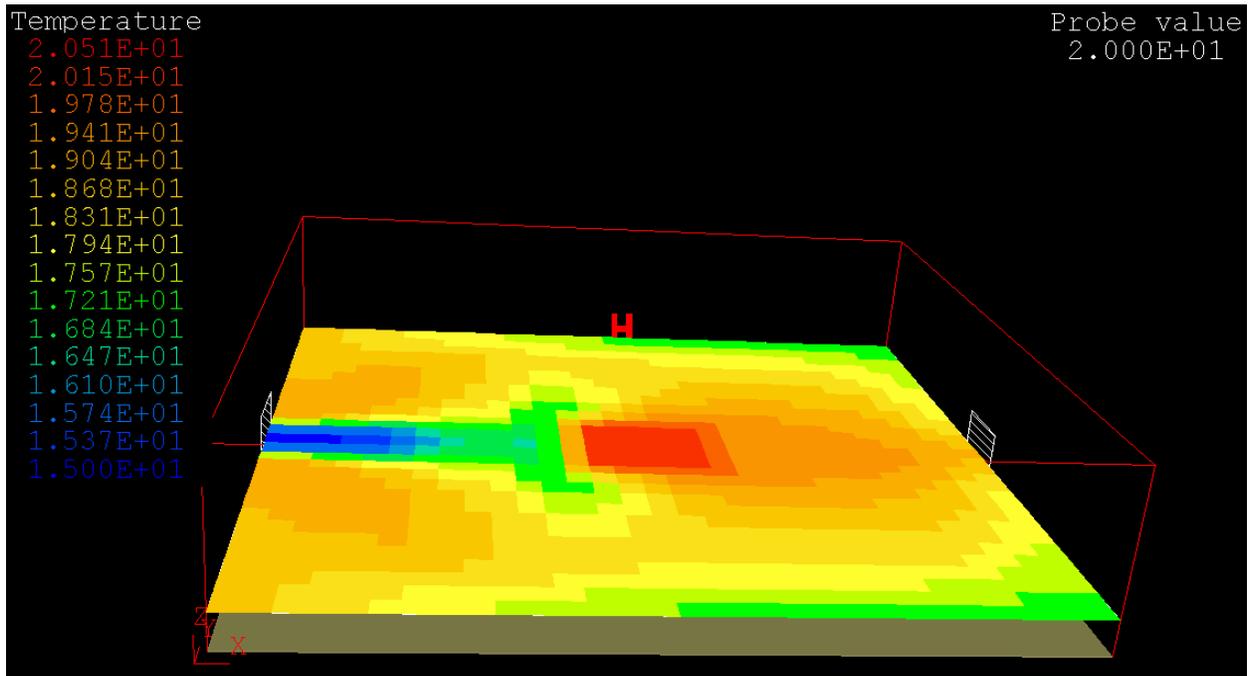
Case 3: Velocity distribution on the horizontal plane $z = 1$



Case 3: Velocity distribution on the vertical plane $y = 10\text{m}$



Case 3: Temperature distribution on the horizontal plane $z = 1$



Case 3: Temperature distribution on the vertical plane $y = 10$

