

2D Odour/gas Plume Simulation Using Fluent CFD

A Technique Report

By Zhenzhang Liu PhD, Post-doc

Harbin Institute of Technology Shenzhen Graduate School,
Dept. of Mechanical Engineering & Automation, China

The University of Adelaide,
School of Mechanical Engineering, Australia

Email: jackydick@hotmail.com
2008-9-1

Confidential

Reprinted is not allowed by lay without the author's permission.

Table of Contents

1	Introduction.....	3
2	Simulation Preparation Using Gambit	4
2.1	Geometry Definition	4
2.2	Specify Boundary Conditions	10
2.2.1	Specify Airflow Inlet and Airflow Outflow.....	10
2.2.2	Specify the Source	11
2.3	Choose Solver and Export Mesh File	13
3	Odour/gas Plume Simulation in Fluent.....	15
3.1	Import Mesh File, Check Grids and Specify Appropriate Unit	15
3.2	Choose Simulation Solver.....	16
3.3	Specify Material Properties.....	19
3.4	Set Boundary Conditions	23
3.5	Set Execute Commands	26
3.6	Run the Simulation	27
3.7	View Simulation Result	28
3.8	Animating the Solution	30
3.9	Data Structure of the Simulation Results in ASCII Format Files	33
4	Customized Boundary Conditions	34
4.1	User Defined Functions (UDFs)	34
4.2	Create UDFs in Fluent	34
4.3	Apply Customized Boundary Conditions Using UDFs	37
4.4	Execute the Simulation and View the Simulation Results.....	38
5	Simulation Data Post-Processing Using OpenDX.....	42
5.1	Get Started	42
5.2	Construct a Data Visualization Program in OpenDX	43
5.3	Parameter Settings for the Modules	45
5.3.1	Parameter Settings for the Format Module	45
5.3.2	Parameter Settings for the Import Module.....	46
5.3.3	Parameter Settings for the Colormap Module	47
5.3.4	Parameter Settings for the Sequencer Module.....	48
5.4	Execute the Data Visualization Program	49
	Reference	51

1 Introduction

This technique report demonstrates using Fluent, which is one kind of Computational Fluid Dynamics (CFD) software to simulate odour/gas plume in a 2D environment. There was a source located in a 2D environment and it released odour/gas at a constant rate in the air. The odour/gas was blown by the airflow and a gas/odour plume was formed under the influence of the airflow. The odour/gas in this report was hydrogen-sulphide (H_2S). The geometry details of the simulation domain are illustrated in Figure 1.1. In Figure 1.1, the airflow entered into the left-hand side boundary of the domain and existed from the right-hand side boundary of the domain.

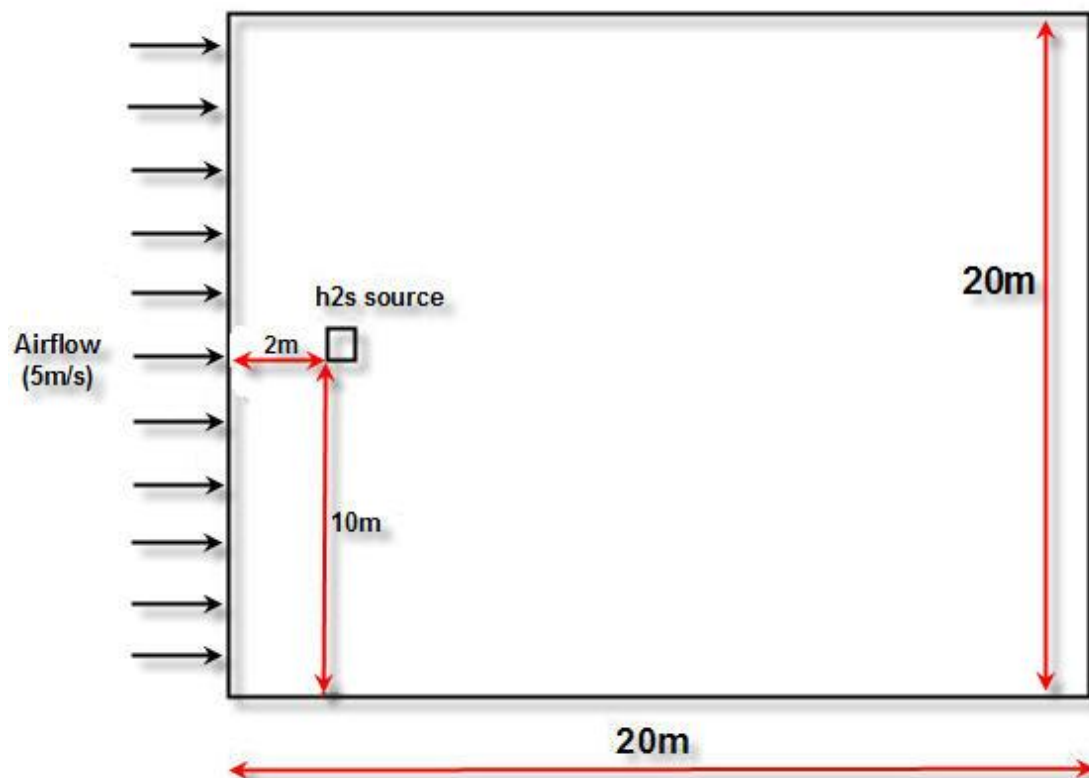


Figure 1.1 Simulation scenario (unit: meter (m))

2 Simulation Preparation Using Gambit

In this section, simulation preparation using Gambit are introduced. Gambit is a pre-processor for Fluent was used to define the geometry of the simulation domain. In Gambit, the simulation domain was constructed and meshed into a number of cells for further simulation in Fluent. Boundary conditions for the simulation domain were also defined in Gambit.

2.1 Geometry Definition

In Gambit, the simulation domain was constructed through the following steps:

- (a) Generate geometry key points of the domain
- (b) Connect the geometry key points to form the domain skeleton
- (c) Generate simulation domain faces
- (d) Mesh the simulation domain faces into a number of cells

(a) Generate geometry key points of the domain

Illustrated in Figure 2.1, in the **Operation Toolpad**, select **Geometry Command Button (1)** >> **Vertex Command Button(2)** >> **Create Vertex Button (3)** then input the key point coordinates in the **Form Filed (4)** and press **Apply(5)** to generate a series of key points. Coordinates of the geometry key points are given in Table 1 and Figure 2.2 demonstrates all the key points generated at this step.

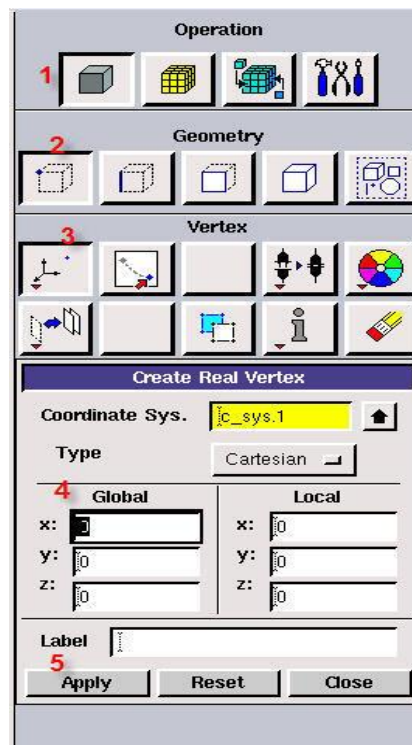


Figure 2.1 Input steps to generate key points

Table 1 Coordinates of Key Points

Key Point Label	X-coordinate	Y-coordinate
1	0	0
2	2	0
3	2.04	0
4	20	0
5	20	20
6	2.04	20
7	2	20
8	0	20
9	2	10
10	2.04	10
11	2	10.04
12	2.04	10.04

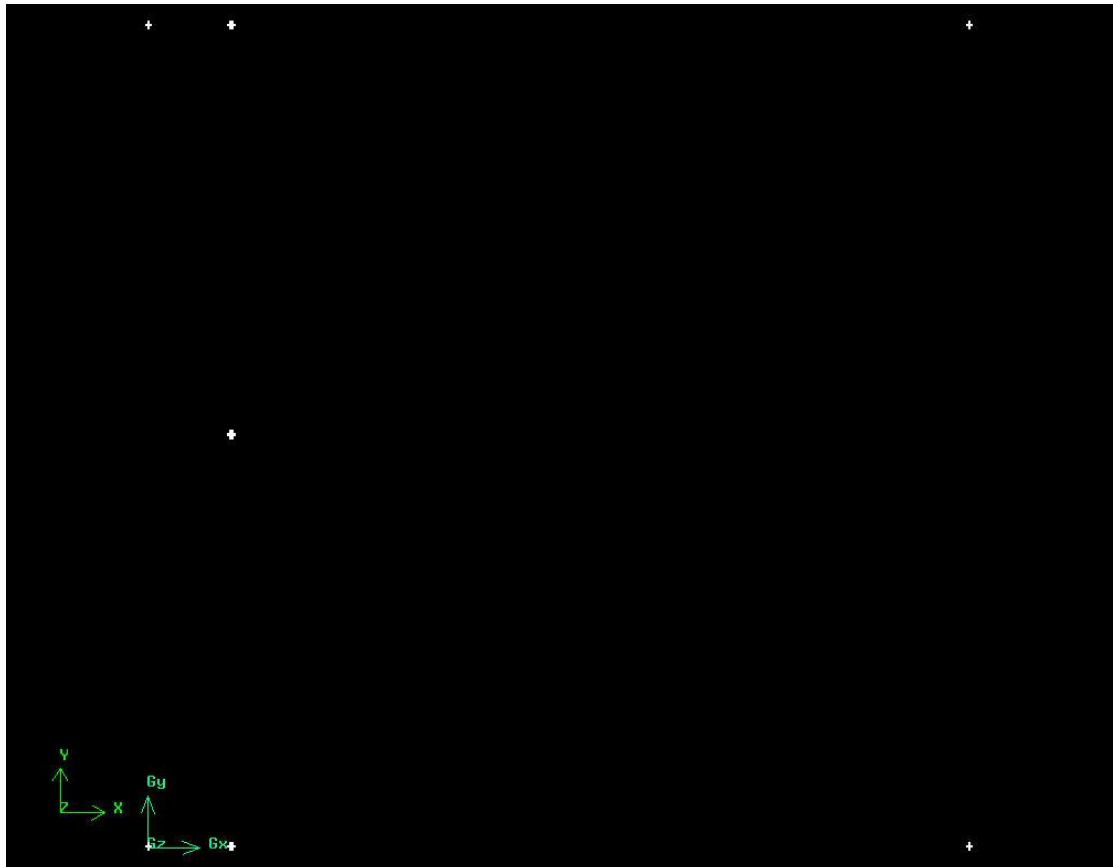


Figure 2.2 Key points generated at step (a). Key points are indicated in white colour points.

(b) Connect the geometry key points to form the domain skeleton

Follow the steps illustrated in Figure 2.3 to connect the key points. In the **Operation Toolpad**, select **Geometry Command Button (1)** >> **Edge Command Button (2)** >> **Create Edge Command Button (3)**. Then press hot keys “**Shift+ mouse left click**” to choose any two key points and press **Apply (4)** to generate domain skeleton. A completed domain skeletons is illustrated in Figure 2.4.

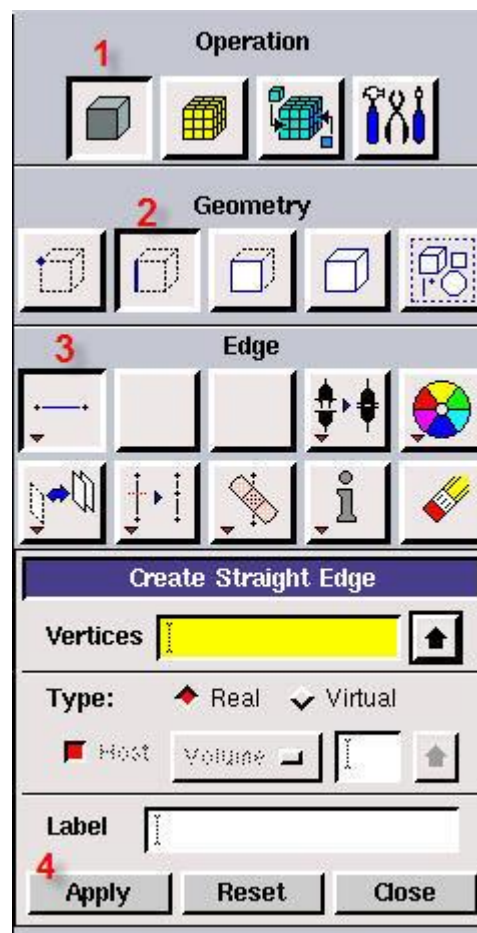


Figure 2.3 Steps to connect key points

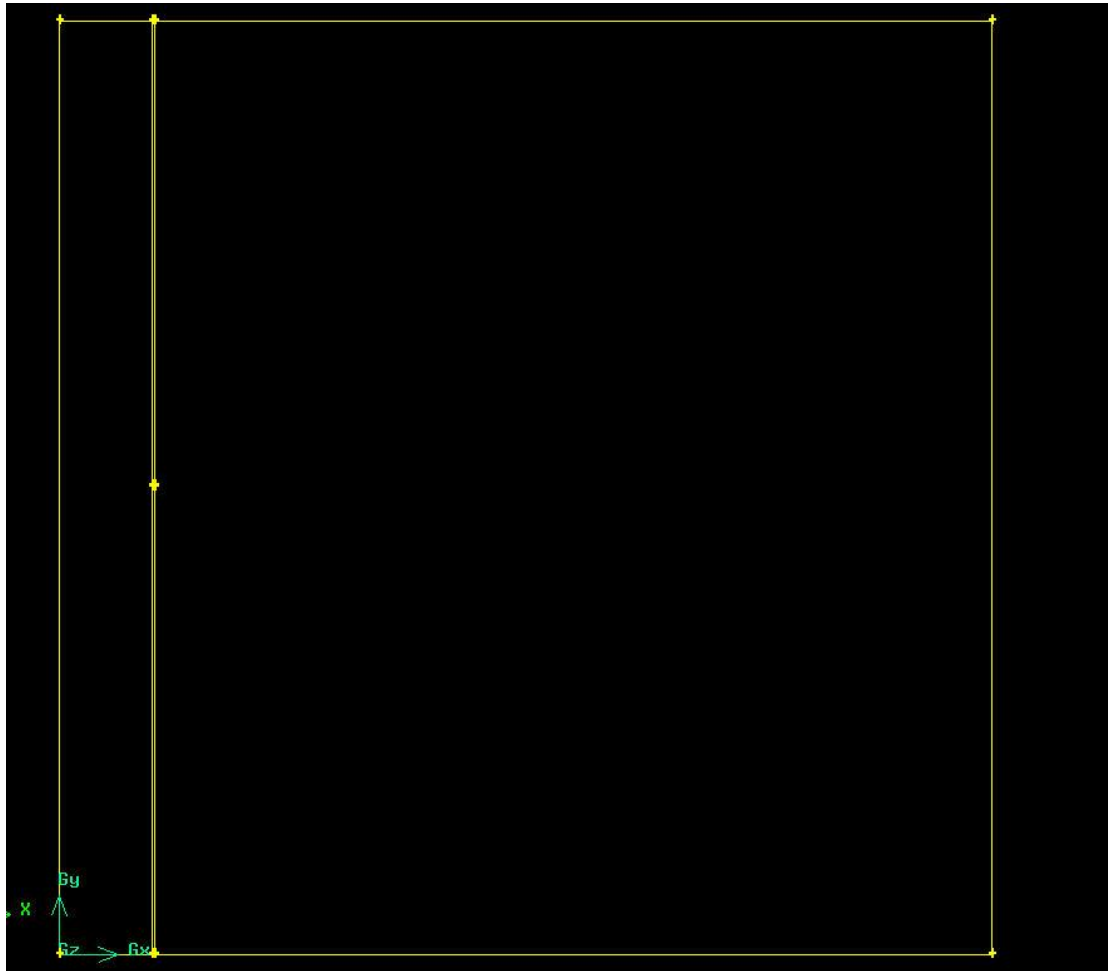


Figure 2.4 Simulation domain skeleton

(c) Generate simulation domain faces

Follow the steps illustrated in Figure 2.5 to generate simulation domain faces. In the **Operation Toolpad**, select **Geometry Command Button (1)** >> **Face Command Button (2)** >> **Form Face (3)**, then press hot keys “**Shift + mouse left click**” to choose appropriate edges and press **Apply (4)** to form faces of the simulation domain. Outcomes of this step are demonstrated in Figure 2.6 and face **S** is the position where an odour/gas source locates.

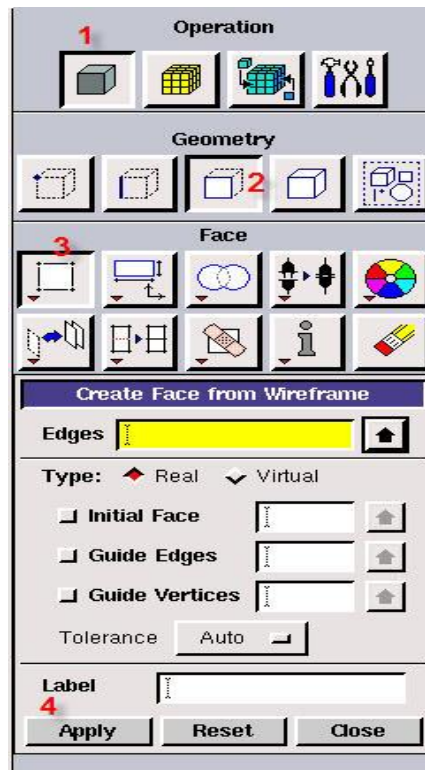


Figure 2.5 Steps to generate simulation domain faces

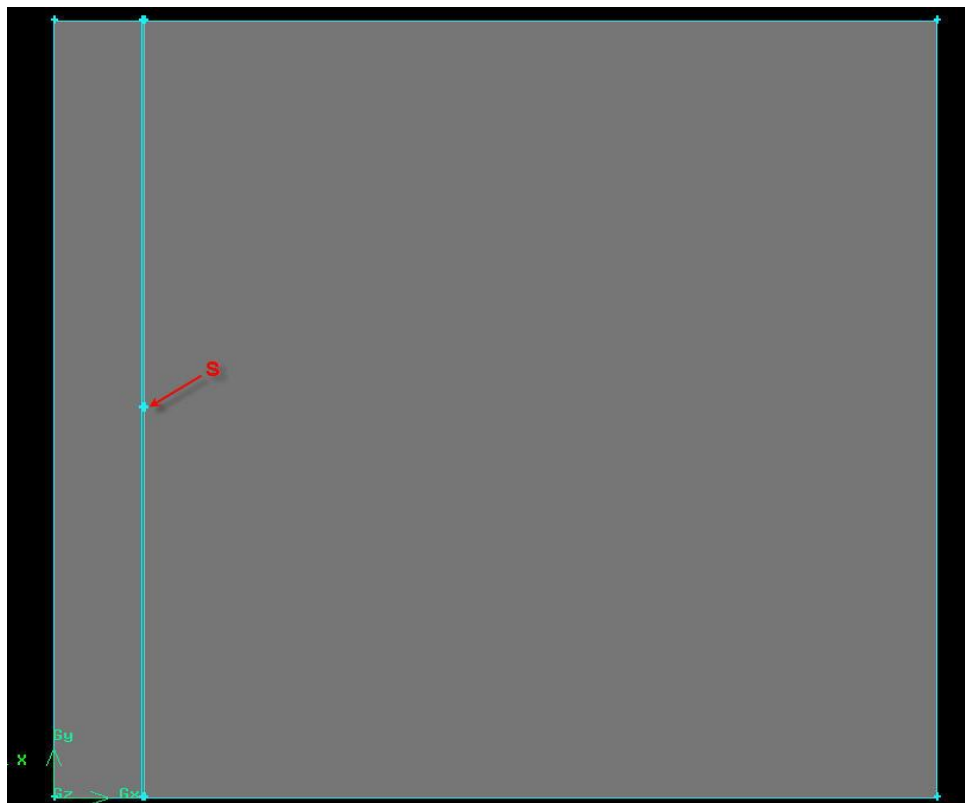


Figure 2.6The simulation domain faces

(d) Mesh the simulation domain

Follow the steps illustrated in Figure 2.7 to mesh the simulation domain. In the **Operation Toolpad** , select **Mesh Command Button (1)** >> **Face Command Button (2)** >> **Mesh Faces (3)** >> **Choose Faces (4)**, choose all faces >> **Specify Mesh Spacing (4)** >> **Apply button (5)**. The complete meshed simulation domain is demonstrated in Figure 2.8. At this step, all the geometry definition had been completed and boundary conditions of the simulation domain were specified in Section 2.2.

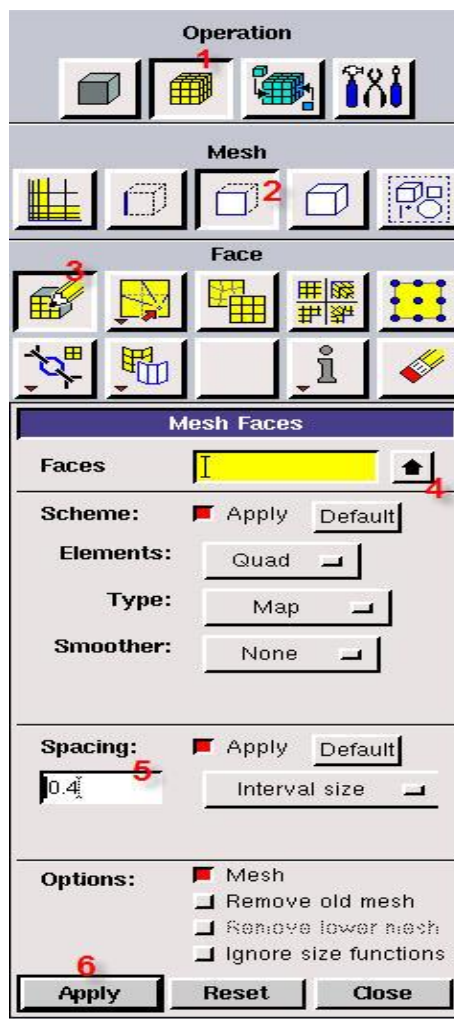


Figure 2.7 Steps to mesh the simulation domain

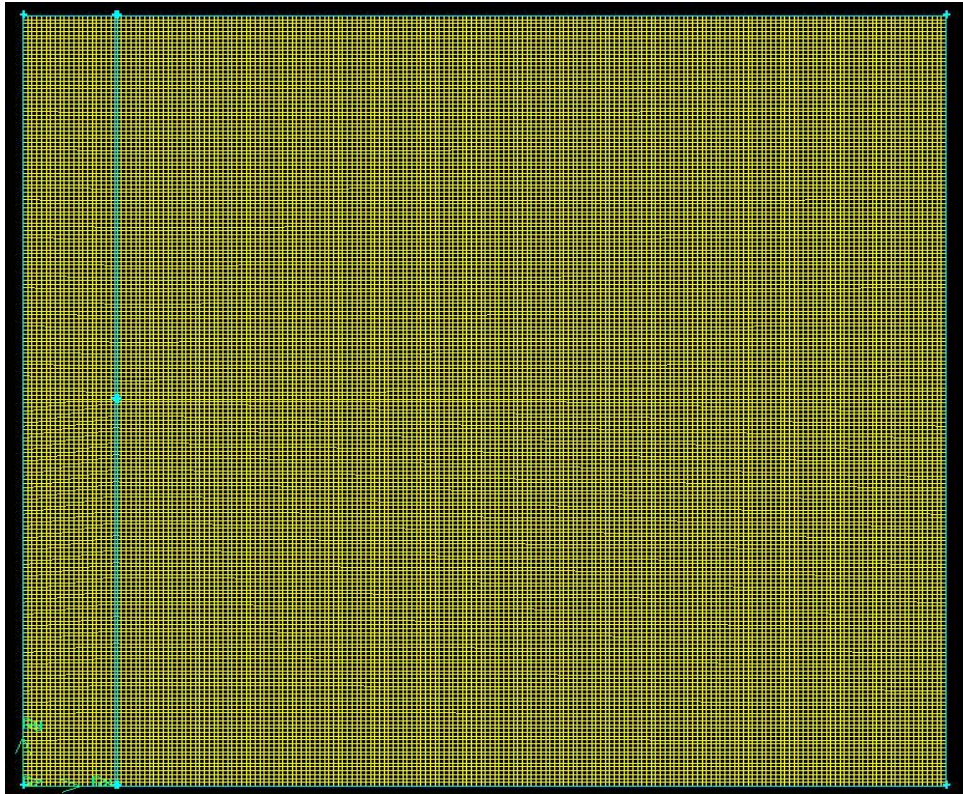


Figure 2.8 The completed meshed simulation domain

2.2 Specify Boundary Conditions

At this section, boundary conditions of the simulation domain are specified. The boundary conditions include: airflow inlet, airflow outlet and source.

2.2.1 Specify Airflow Inlet and Airflow Outflow

The airflow inlet is the left-hand side boundary of the simulation domain and the airflow outlet is the right-hand side boundary of the simulation domain. In the **Operation Toolpad**, follow the steps illustrated in Figure 2.9 to specify airflow inlet and outflow. Select **Zones Command Button (1)** >> **Specify Boundary Types Command Button (2)** >> press **Add (3)** >> give a name for the boundary (4) >> select **boundary type (5)** >> use hot keys “**shift + mouse left click**” to select appropriate boundary >> press **Apply (6)**. Similar approach was adopted to specify the airflow outlet boundary condition. The boundary type for the airflow inlet is **velocity_inlet** and the boundary type for the airflow outlet is **outflow** in this report.

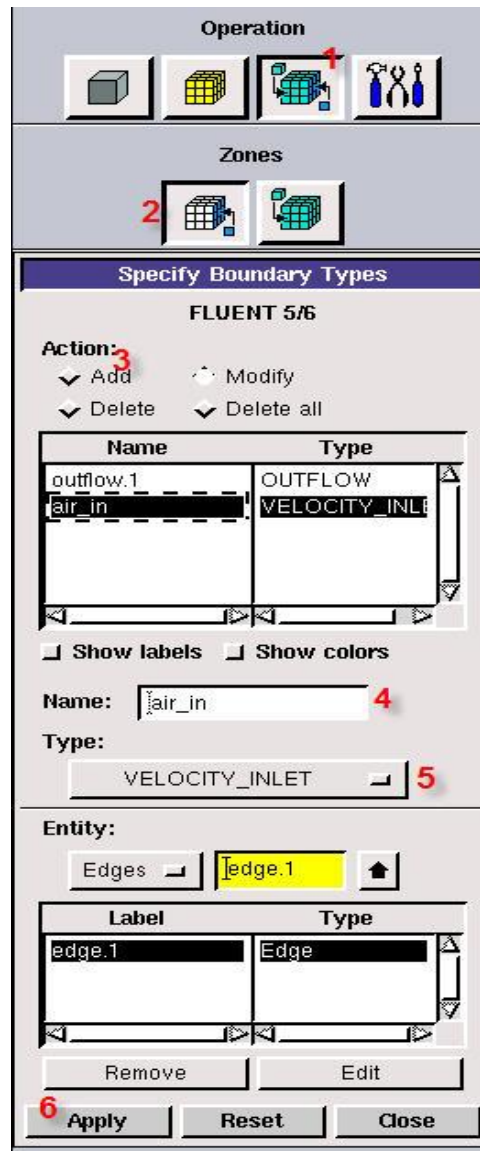


Figure 2.9 Steps to specify boundary conditions

2.2.2 Specify the Source

Follow the steps illustrated in Figure 2.10 to specify the location of the odour/gas source. In **Operation Toolpad**, select **Zones Command Button (1)** >> **Specify Continuum Types Command Button (2)** >> press **add (3)** >> give a name for the source **(4)** >> **select the appropriate continuum type (5)**, here “Fluid” type is chosen >> use hot keys “**shift + mouse left click**” to choose the face where a source was located as illustrated in Figure 2.6 >> press **Apply (6)** to complete the operation.

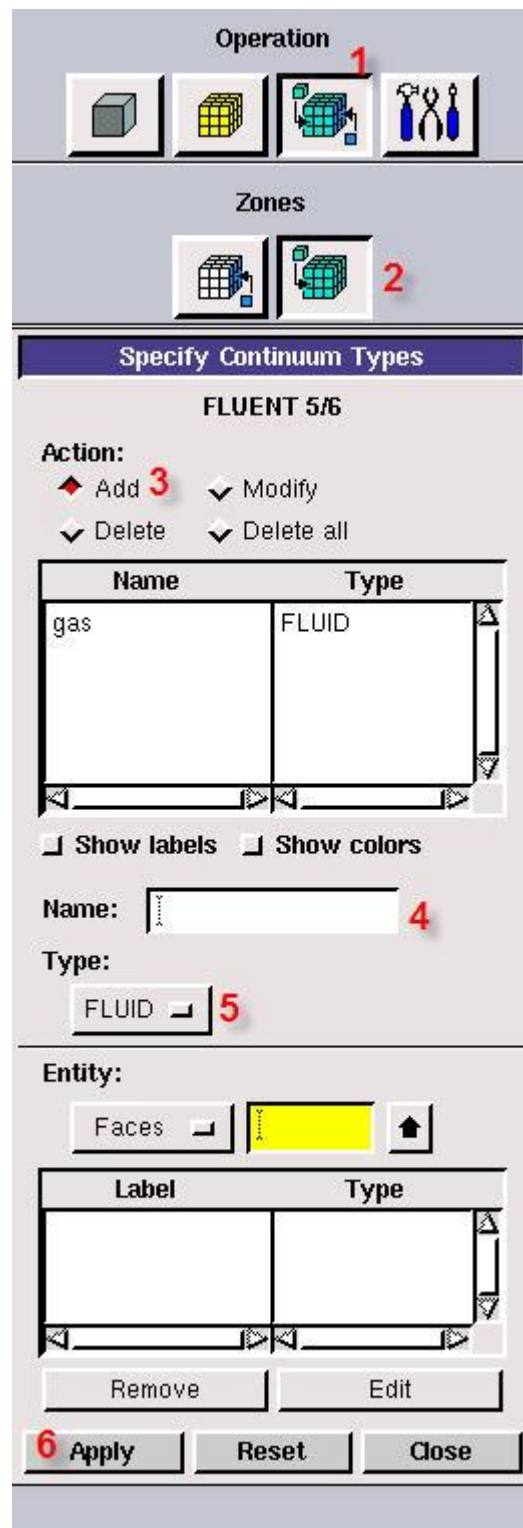


Figure 2.10 Steps to specify an odour/gas source in the simulation domain

2.3 Choose Solver and Export Mesh File

As the geometry of the simulation domain was imported into Fluent for further simulation, therefore a solver should be specified in Gambit. In menu bar, select **Solver** then choose **Fluent 5/6** as illustrated in Figure 2.11.



Figure 2.11 Specify a solver for further simulation.

Then export the geometry file as a mesh file which can be imported into Fluent. In menu bar select **File**>> **Export** >> **Mesh** as illustrated in Figure 2.12. A file **Export File Panel** will appear then specify a name for the mesh file and choose the “**Export 2-D(X-Y) Mesh**” option as illustrated in Figure 2.13 then click **Accept** to export the mesh file under the name specified.

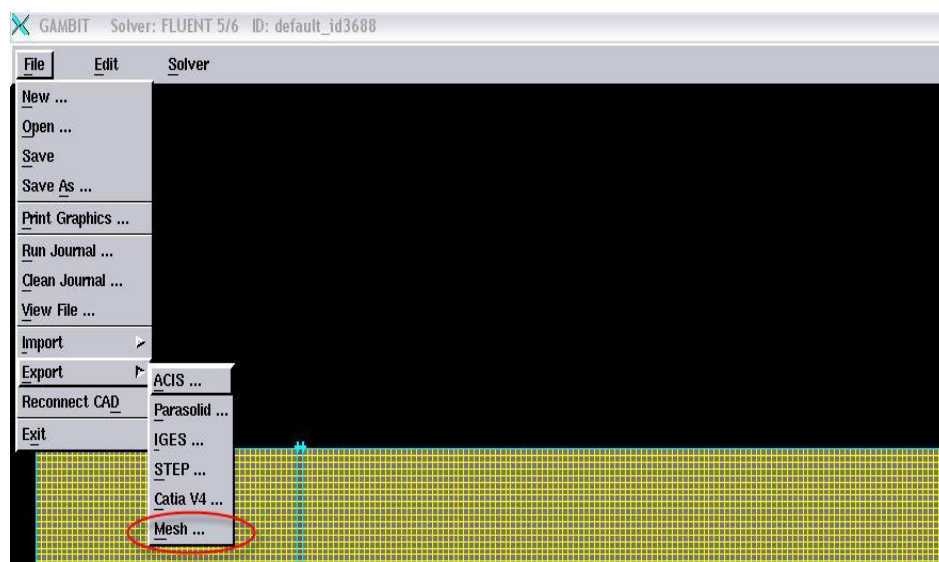


Figure 2.13 Export the geometry file as a mesh file

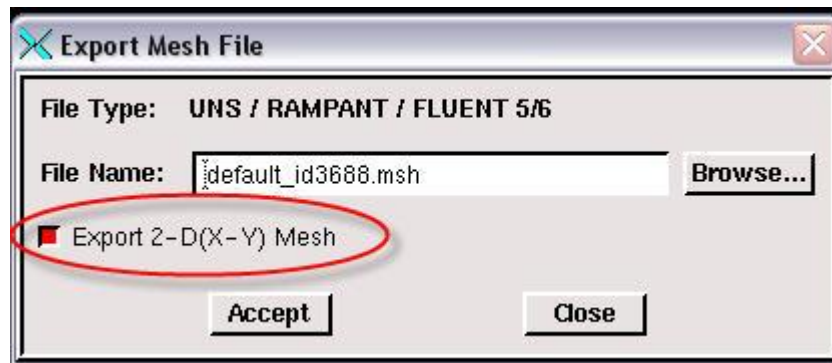


Figure 2.13 Export Mesh File panel

3 Odour/gas Plume Simulation in Fluent

The mesh file from Gambit can be imported into Fluent for further simulation. In this section the simulation process in Fluent are explained in detail.

3.1 Import Mesh File, Check Grids and Specify Appropriate Unit

Run Fluent software, in menu bar select **File >> Read >> Case**, then choose the mesh file generated by Gambit. Then select **Grid >> Check** in menu bar to check the imported mesh file. If the file is damaged, it is necessary to fix it using Gambit. In menu bar select **Grid >> Scale**, a **Scale Grid** panel will appear as illustrated in Figure 3.1. In the **Scale Grid** panel, choose the preferred unit in **Unit Conversion** illustrated as step 1 in Figure 3.1. In this report, the unit was chosen as meter (m). Then click **Scale** (step 2 in Figure 3.1) only once and close the panel.

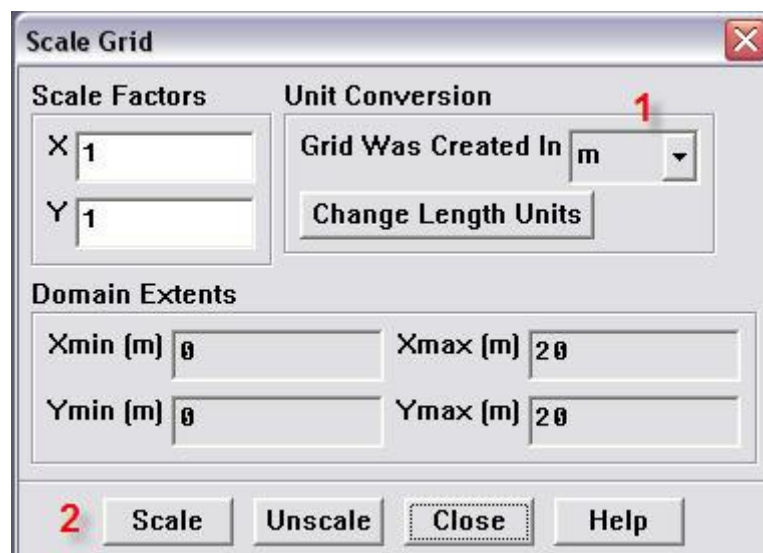


Figure 3.1 Scale Grid Panel

3.2 Choose Simulation Solver

Fluent provides two solvers for plume simulation: the segregated solver and the coupled solver. These two solvers have different approaches to solve the governing integral equations for the conservation of mass, momentum and chemical species. In this report, a point source was located in a simulated domain and it releases gas/odour to form a plume in the air, therefore it is a fluid species transport (without chemical reaction) problem. It is assumed that there were only two kinds of fluid species existed in the simulated environment, which were the gas/odour and the air. In this simple situation, the segregated solver was chosen to simulate gas/odour plume. In menu bar, select **Define >> Model >> Solver**, then specify appropriate values as illustrated in Figure 3.2.

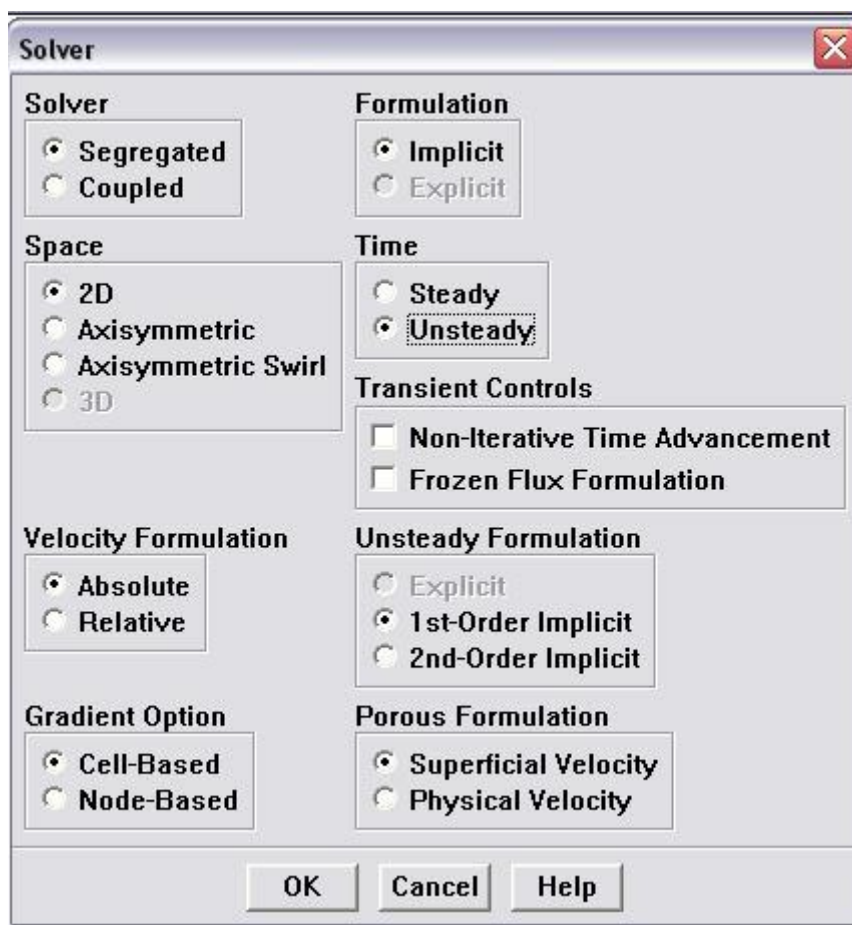


Figure 3.2 Solver Selection

As plume simulation is a species transport problem, therefore we should also enable Fluent activate the species-transport model. In tool bar, select **Define >> Model >> Species >> Transport & Reaction** and specify appropriate values as illustrated in Figure 3.3. It is also necessary to enable the solver handles turbulence problems, therefore in menu bar select **Define >> Model >> Viscous** and accept default settings as illustrated in Figure 3.4.

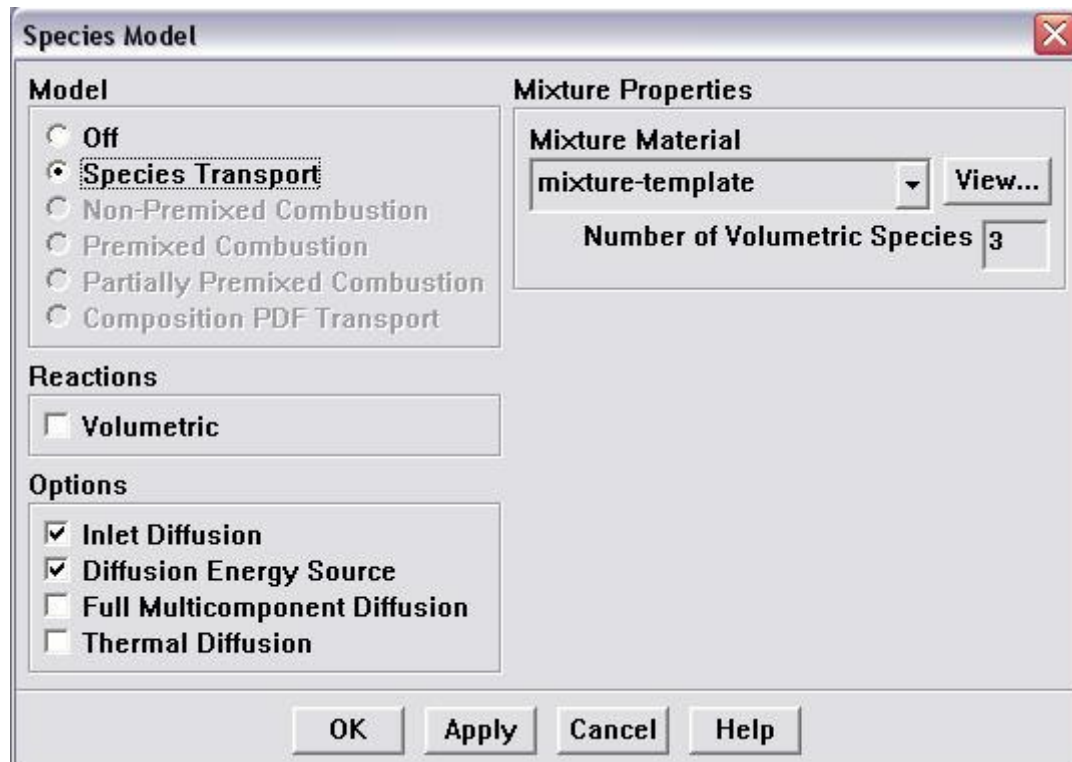


Figure 3.3 Enable species transport model

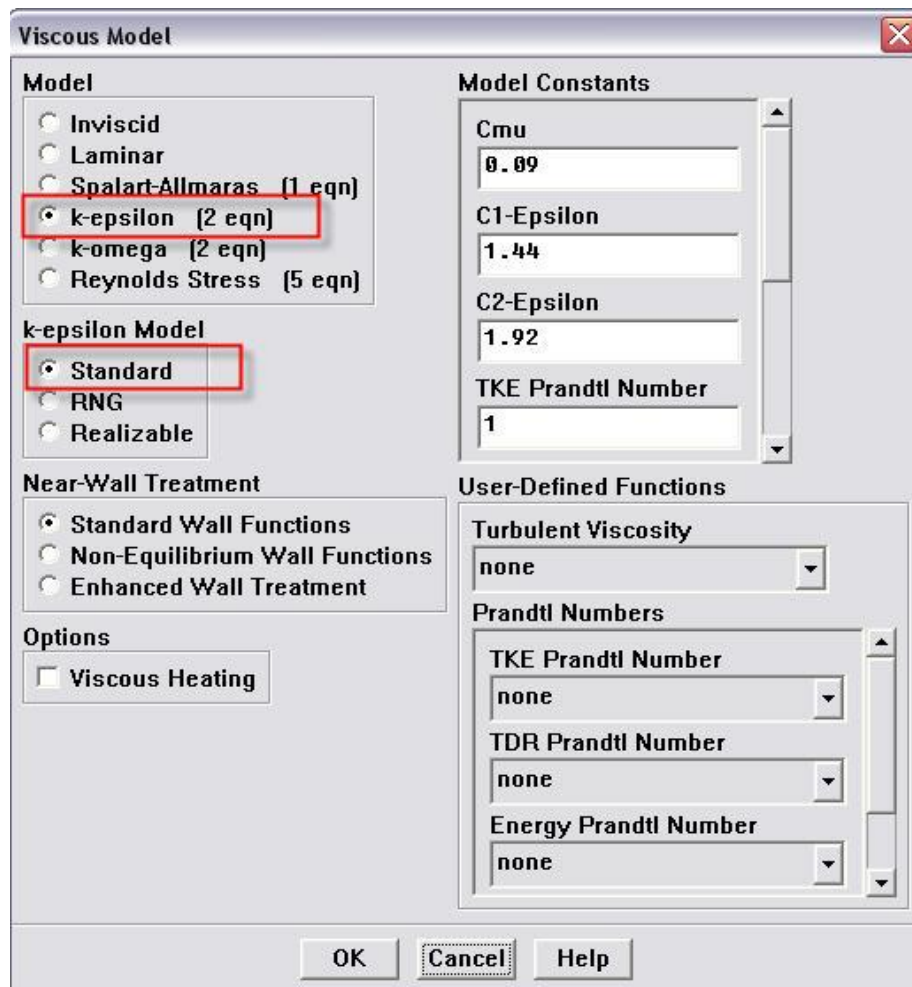


Figure 3.4 Viscous Model Setting Panel

3.3 Specify Material Properties

In this report, there are only two kinds of species which are the air and hydrogen-sulphide (H_2S). In Fluent, air is one kind of default material and we just need to specify properties of H_2S . In menu bar, select **Define >> Material**, a **Material** panel illustrated in Figure 3.5 will appear. Then select **Fluent Database** to select properties of H_2S . A **Fluent Material Database** illustrated in Figure 3.6 appears and select the appropriate material as demonstrated in Figure 3.6 and press **Copy** button return to **Material Panel**, in **Material Panel** press **Change/Create** button to complete the material selection operation.

The screenshot shows the 'Materials' panel in ANSYS Fluent. The panel is titled 'Materials' and has a close button (X) in the top right corner. It is divided into several sections:

- Name:** 'hydrogen-sulfide'
- Material Type:** 'fluid' (dropdown menu)
- Chemical Formula:** 'h2s'
- Fluent Fluid Materials:** 'hydrogen-sulfide (h2s)' (dropdown menu)
- Mixture:** 'none' (dropdown menu)
- Order Materials By:** 'Name' (radio button selected), 'Chemical Formula' (radio button unselected)
- Fluent Database...** (button, circled in red)
- User-Defined Database...** (button)
- Properties:**
 - Density (kg/m3):** 'constant' (dropdown menu), '1.46' (text input), 'Edit...' (button)
 - Cp (J/kg-k):** 'constant' (dropdown menu), '1170' (text input), 'Edit...' (button)
 - Thermal Conductivity (W/m-k):** 'constant' (dropdown menu), '0.0134' (text input), 'Edit...' (button)
 - Viscosity (kg/m-s):** 'constant' (dropdown menu), '1.2e-05' (text input), 'Edit...' (button)
- Buttons:** 'Change/Create' (button, circled in red), 'Delete', 'Close', 'Help'

Figure 3.5 Material Panel

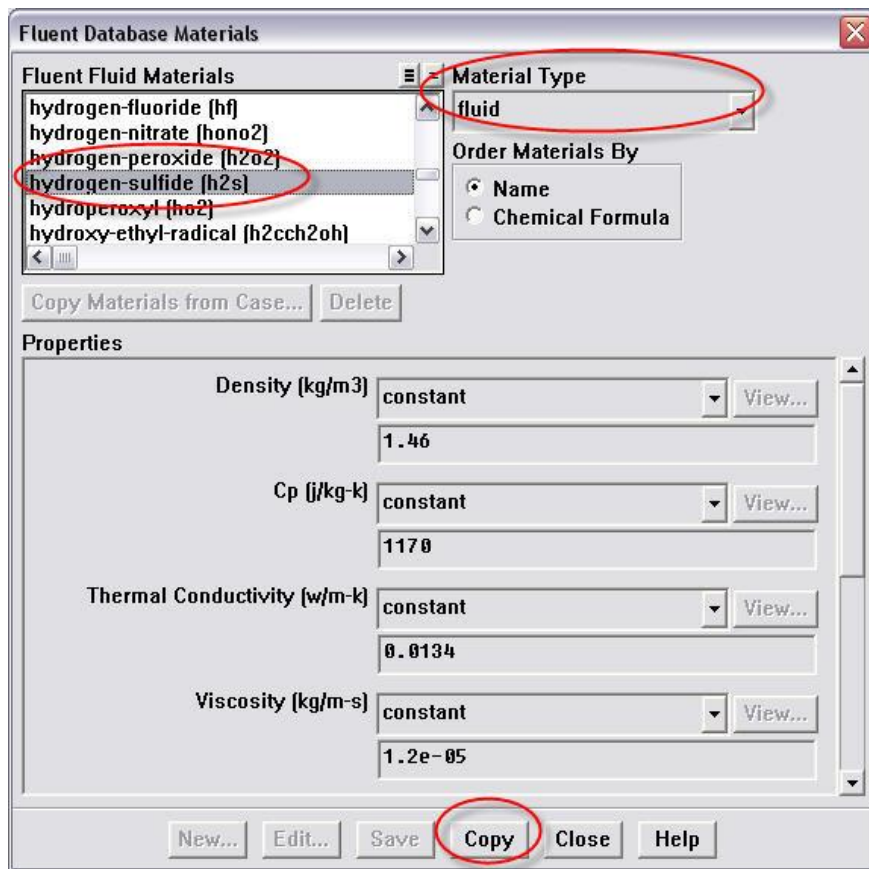


Figure 3.6 Fluent Material Database Panel

To ensure appropriate materials have been selected, in tool bar select **Define >> Model >> Species >> Transport & Reaction**, a **Species Model** panel appears and press the **Edit** button as illustrated in Figure 3.7. A **Material** panel appears then press the **Edit** button in the **Material** panel as illustrated in Figure 3.8. Ensure the selected materials in the **Species** panel are the same as illustrated in Figure 3.9.

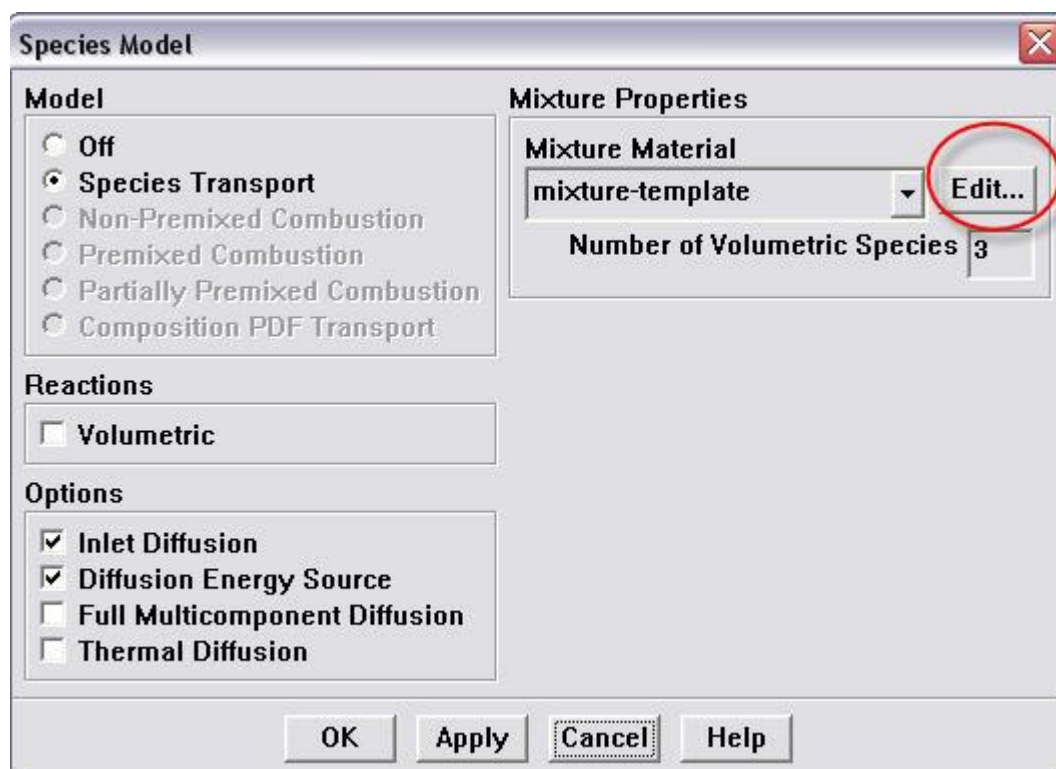


Figure 3.7 Species Model Panel

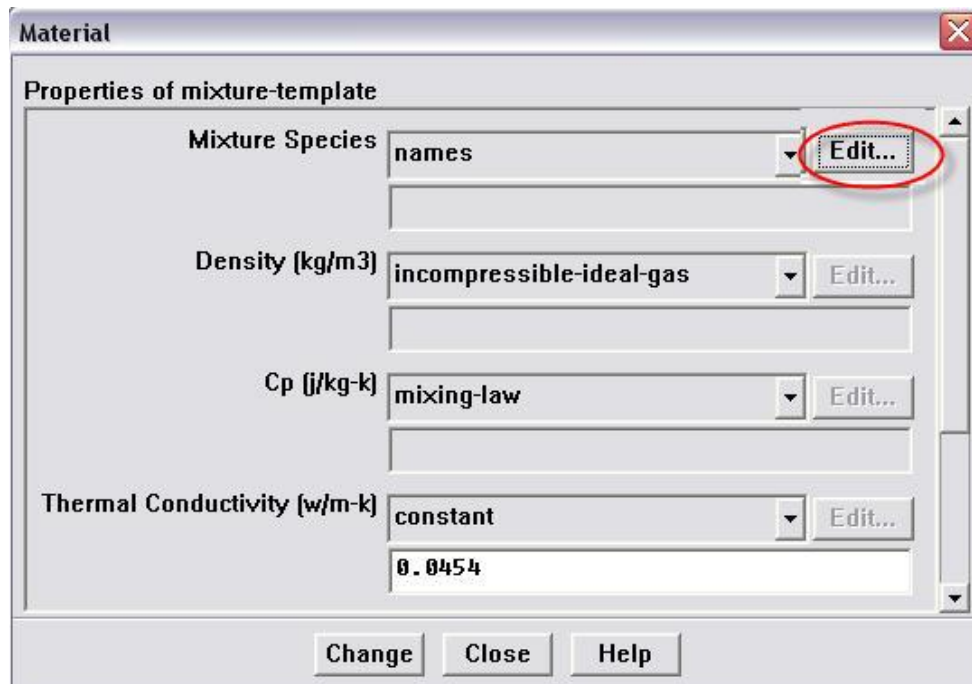


Figure 3.8 Material Panel

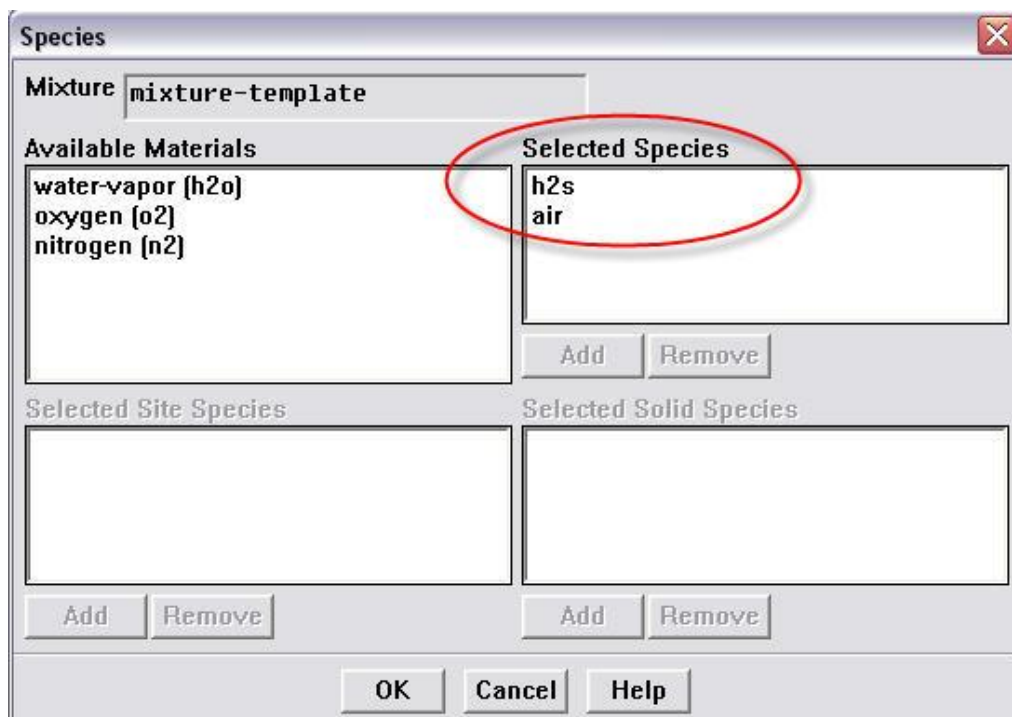


Figure 3.9 Species Panel

3.4 Set Boundary Conditions

Boundary conditions include the airflow inlet velocity, airflow outlet and the source release rate. Firstly, the airflow inlet velocity was specified. In menu bar, select **Define >> Boundary Conditions**, a **Boundary Conditions** panel will appear and is illustrated in Figure 3.10. In the **Zone** column, the boundary names specified in Gambit can be found. Here **air_in** is the name of the airflow inlet, **outflow.2** is the name of the airflow outlet boundary and **fluid.3** is the name of the source. Select **air_in** in the **Zone** column and select the correspondent boundary type **velocity-inlet** in the **Type** column then press the **set** button. A **Velocity Inlet** panel will appear and the airflow velocity enters into the simulation domain can be specified. The **Velocity Inlet** panel is illustrated in Figure 3.11. Fluent will handle the airflow outlet boundary conditions; therefore it is not necessary to set airflow outlet boundary conditions manually.

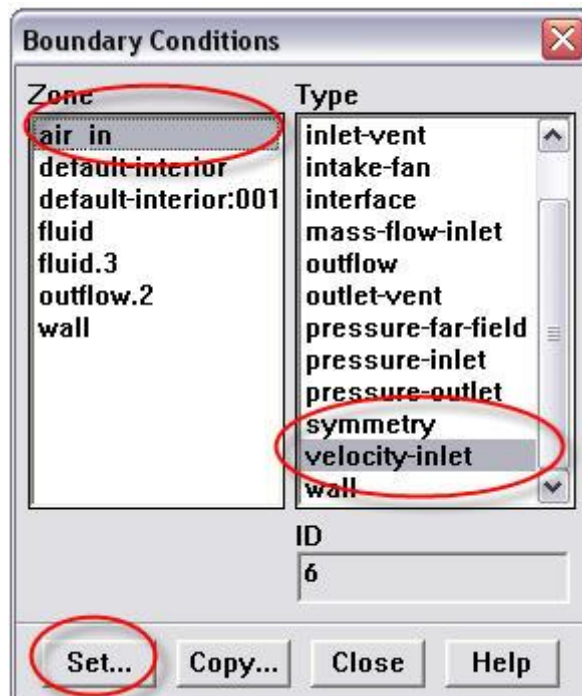


Figure 3.10 Boundary Conditions Panel

Velocity Inlet

Zone Name:

Velocity Specification Method: **Magnitude, Normal to Boundary**

Reference Frame: **Absolute**

Velocity Magnitude [m/s]: **constant**

Temperature [K]: **constant**

Turbulence Specification Method: **K and Epsilon**

Turb. Kinetic Energy [m²/s²]: **constant**

Turb. Dissipation Rate [m²/s³]: **constant**

Species Mass Fractions

Species	Mass Fraction	Value	Unit	Specification
h2s	<input type="text" value="0"/>	0		constant

OK Cancel Help

Figure 3.11 Velocity Inlet Panel

In the **Boundary Conditions** panel, select the source term namely **fluid.3** and select **fluid** in the type column then press the **set** button in the **Fluid** panel and specify the parameters as illustrated in Figure 3.12. In the Fluid panel the release rate of H₂S is 500 kg/m³-s and the total mass of the species (H₂S +air) in the simulation domain is 1000 kg/m³-s. Press **OK** to accept the parameter settings.

Fluid

Zone Name
fluid.3

☒ Source Terms
☐ Fixed Values
☐ Porous Zone

Motion Source Terms Fixed Values Porous Zone Reaction

Mass [kg/m ³ -s]	1000	constant
X Momentum (n/m ³)	10	constant
Y Momentum (n/m ³)	2	constant
h2s [kg/m ³ -s]	500	constant

OK Cancel Help

Figure 3.12 Fluid Panel

3.5 Set Execute Commands

During the simulation, simulation data for every time step need to be saved in ASCII files, therefore we should specify commands to let Fluent save every time step simulation results in files. In menu bar, select **solve >> execute commands** and input appropriate commands in the **Execute Commands** panel as illustrated in Figure 3.13.

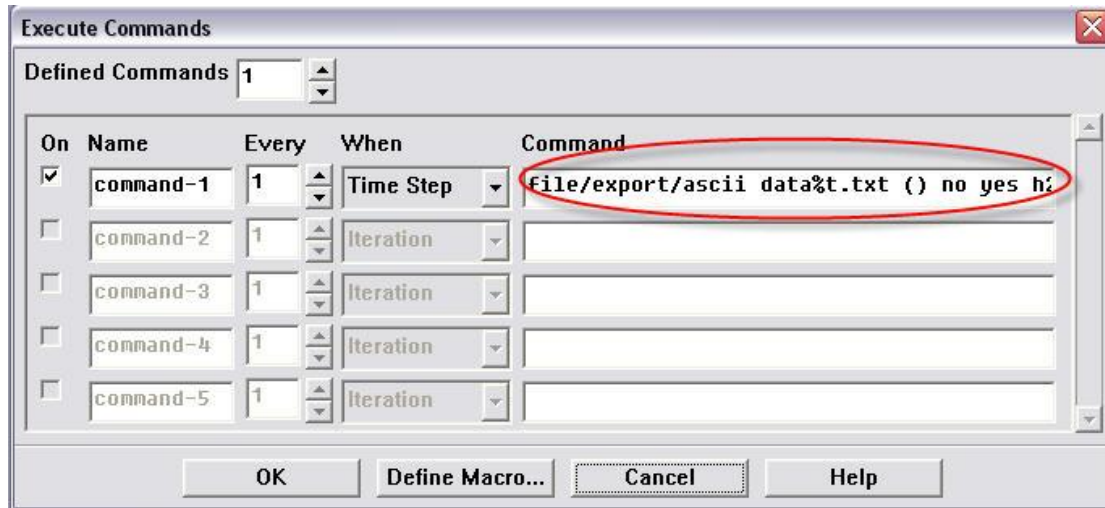


Figure 3.13 Execute Commands Panel

Commands in the command file are:

file/export/ascii data%t.txt () no yes h2s y-velocity x-velocity q no

The above commands control Fluent to save the airflow velocities (x-velocity, y-velocity) and the concentration of H₂S at every grid in the simulation domain in ASCII format files during the simulation. The file name of the ASCII format file is:

data+time_step_number.txt

3.6 Run the Simulation

In menu bar select **solve >> iterate**, specify the number of simulation time steps then press **iterate** button to execute the simulation. The **Iterate** panel is illustrated in Figure 3.14.

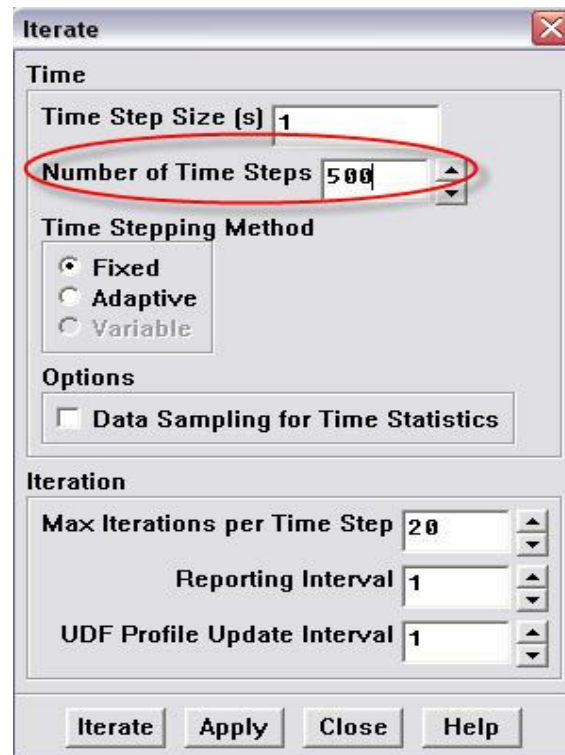


Figure 3.14 Iterate Panel

3.7 View Simulation Result

After the simulation was completed, Fluent is able to visualize the simulation results. In menu bar, select **display >> contours**, a **Contours** panel appears and select appropriate options as illustrated in Figure 3.15 and the H₂S plume in the simulation domain is visualized in a separated window as illustrated in Figure 3.16.

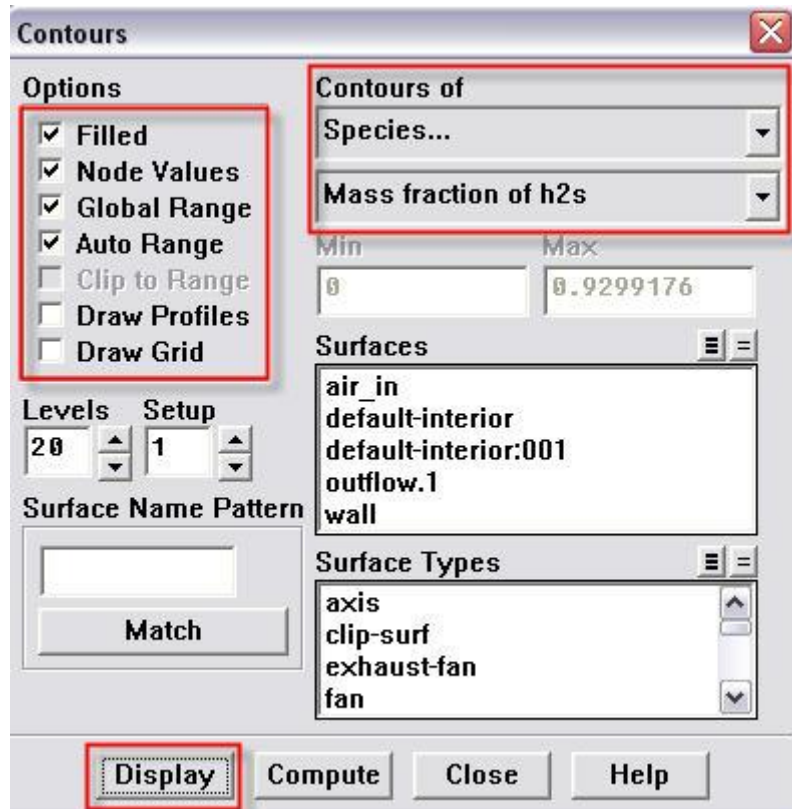


Figure 3.15 Contours Panel

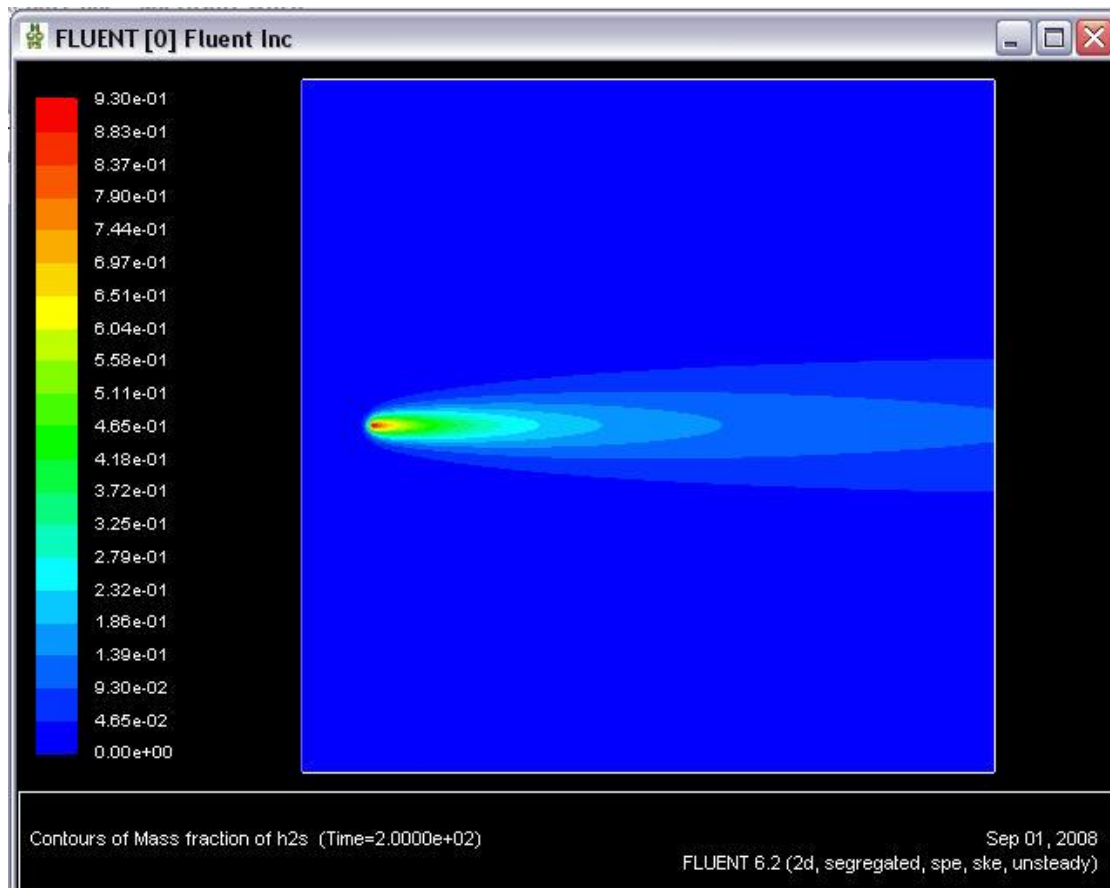


Figure 3.16 H_2S plume in the simulation domain

3.8 Animating the Solution

During the calculation, Fluent can create an animation of contours, vectors, XY plots. Before start the calculation, variables and types of plots will be animated need to be specified. During the simulation, Fluent will display the requested plots and these plots can also be stored as a series of images or movie files.

The procedure to define animation sequence is as follows:

1) In the Fluent menu bar select **Solve >> Animate >> Define**, a **Solution Animation** panel will appear (Figure 3.17). Increase the **Animation Sequences** value to the number of animation sequences you want to specify. Enter a name for the sequence under the **Name** heading. Indicate how often you want to create a new frame in the sequence by setting the interval under **Every** and selecting **Iteration** or **Time Step** in the drop-down list below **When**. Click the **Define** button to open the Animation Sequence panel (Figure 3.18).

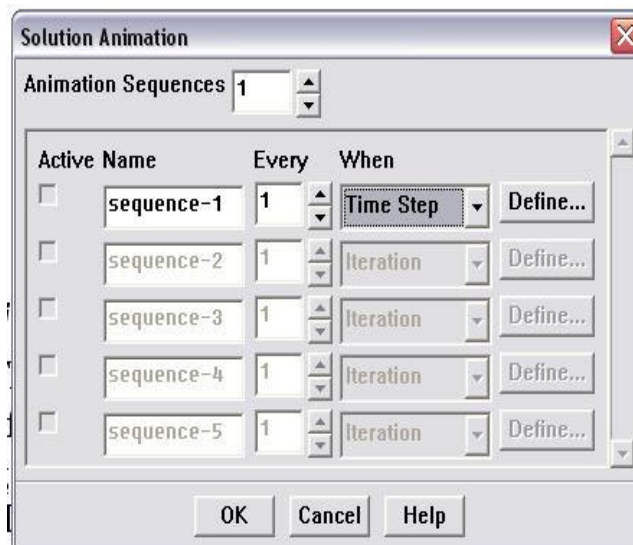


Figure 3.17 The Solution Animation panel

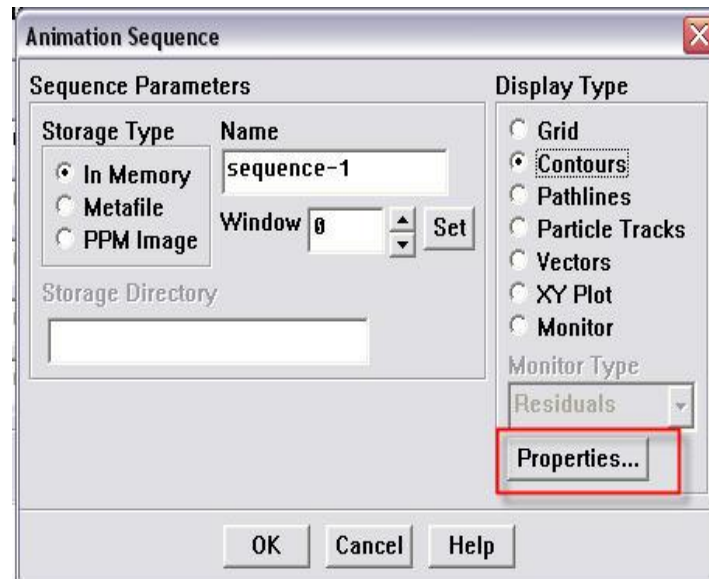


Figure 3.18 The Animation Sequence panel

2) In the **Animation Sequence** panel, specify whether you want FLUENT to save the animation sequence frames in memory or on your computer's hard drive. To save the animation sequence in memory, select **In Memory** under **Storage Type**. To save the animation sequence to your computer's hard drive as a graphics meta file, select **Metafile** under **Storage Type**. To save the animation sequence to your computer's hard drive as a pixmap image, select **PPM Image** under **Storage Type**. In this report, we want Fluent displays the odour/gas source concentration during the simulation, therefore, select **Contours** in the **Display Type** then a **Contour** panel will appear (Figure 3.19).

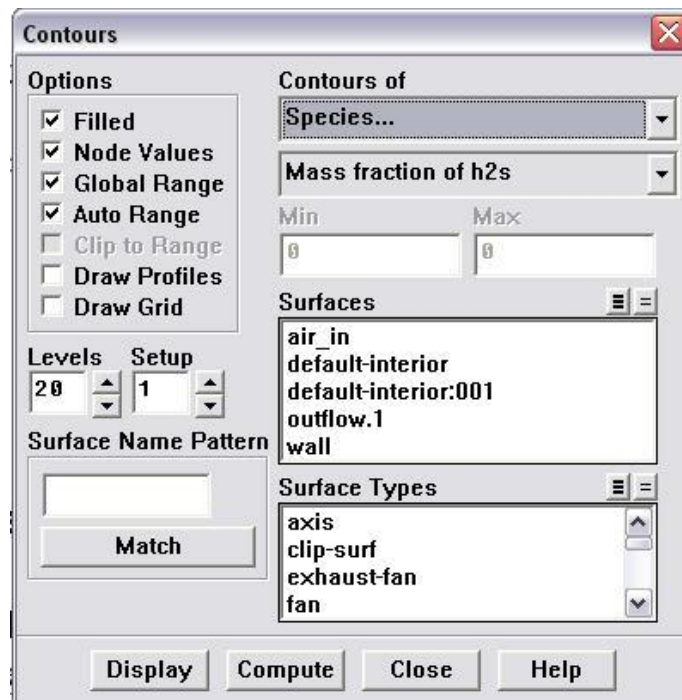


Figure 3.19 The Contour panel

3) In the **Contour** panel, select **Species** and **Mass fraction of h2s** in the **Contours of** section and Fluent will display the h2s concentration contours during the simulation. Then press the **Display** button, a window which will display the requested plots will appear. Press the **Close** button to exit the **Contour** panel, and click **ok** to close the **Animation Sequence** panel and the **Solution Animation** panel.

After the definition of animation sequence, follow the steps given in Section 3.6, we can see the simulation results displayed during the calculation. In this report, we can see how the plume propagates under the influence of the airflow.

3.9 Data Structure of the Simulation Results in ASCII Format Files

In section 3.4, **Execute Commands** were set and Fluent saved simulation results at different time step in ASCII format files. The ASCII file is a text file that contains one header line indicating the names of the variables (eg. x-velocity, y-velocity and H₂S) and followed by the correspondent variable values. The structure of the ASCII file is illustrated in Table2.

Table 2 Illustration of the data structure of ASCII file

Node number	x-coordinate	y-coordinate	x-velocity	y-velocity	H₂S
1	2	10	5.54E-01	-1.50E+00	8.41E-01
2	2	10	5.56E-01	1.51E+00	8.41E-01
3	2.04	10	3.21E+00	1.59E+00	9.30E-01
4	2.04	10	3.21E+00	-1.57E+00	9.30E-01
5	0	0	2.50E+00	0	7.64E-18
6	1	0	0	0	4.17E-17
7	2	0	0	0	1.26E-16
.
.

4 Customized Boundary Conditions

In Section 3.4, when the boundary conditions were specified for the airflow inlet, it can be found that Fluent only provides several standard velocity specification methods for an airflow inlet which are:

- (a) Magnitude, Normal to Boundary. This method specifies the velocity of the airflow and the direction of the airflow is normal to the inlet boundary.
- (b) Components. This method needs the input of the velocity components (e.g. x-velocity component, y-velocity component).
- (c) Magnitude and Direction. The method requires the velocity of the airflow and a fixed direction of the airflow.

The above methods may not be adequate once unstable airflow condition is adopted (e.g. wind shift) in simulation, therefore it is necessary to define user customized boundary conditions using User Defined Functions (UDFs). In this section, the approaches to define user customized boundary conditions are introduced.

4.1 User Defined Functions (UDFs)

A user-defined function, or UDF, is a function that can be dynamically loaded with the FLUENT solver to enhance the standard features of the code. UDFs are written in the C programming language. They are defined using DEFINE macros that are supplied by Fluent Inc. Every UDF contains the **udf.h** file inclusion directive (`#include "udf.h"`) at the beginning of the source code, which allows definitions for DEFINE macros and other Fluent-provided macros and functions to be included during the compilation process. UDFs are written in C using any text editor and the source file is saved with a .c file extension.

4.2 Create UDFs in Fluent

For example, if the airflow entered into the simulation domain was at a constant velocity but shifted between $\pm 22.5^\circ$ (0° indicates the horizontal direction), UDFs can be adopted to define this kind of customized boundary conditions. To define and use a UDF, several steps need to be followed:

(a) Define the problem

Consider the same simulation scenario as illustrated in Figure 1.1, varying airflow directions were adopted instead of the fixed airflow direction. The varying airflow entered into the left-hand side boundary of the simulation domain at a constant 5m/s velocity but varied between $\pm 22.5^\circ$ during the simulation.

Let v_x the x component of airflow,

v_y the y component of airflow,

α the angel between the airflow direction and x horizontal axis, counter clockwise direction is the positive angle direction

There exists the following relation:

$$v_x = 5 \times \cos \alpha$$

$$v_y = 5 \times \sin \alpha$$

$$\alpha \in [-22.5^\circ; 22.5^\circ]$$

(b) Create the UDF source file

Based on the problem defined in (a), two UDFs were created to generate v_x and v_y during the simulation to simulate a wind shift with a constant airflow velocity. These two UDFs are as follows:

```
/*vxprofile.c*/
#include "udf.h"

DEFINE_PROFILE(inlet_x_velocity,thread,position)    // inlet_x_velocity will become a boundary
                                                    // method name
{
    face_t f;
    real t=CURRENT_TIME;
    real alpha;

    alpha = 22.5*sin(10*t*3.14/180);                // whind shift angle varies between
                                                    // [-22.5 ;+22.5 ], parameters in sin function
                                                    // specify the wind shift velocity, here the wind
                                                    // shifts at a velocty of 2 %s

    begin_f_loop(f,thread)
    {
        F_PROFILE(f,thread,position)=5*sin(alpha*3.14/180); // x component of airflow
                                                            // varies due to variation of
                                                            // alpha
    }
    end_f_loop(f,thread)
}
```

```

/*vxprofile.c*/
#include "udf.h"

DEFINE_PROFILE(inlet_y_velocity,thread,position)    // inlet_y_velocity will become a
                                                    // boundary method name
{

    face_t f;
    real t=CURRENT_TIME;
    real alpha;

    alpha = 22.5*sin(10*t*3.14/180);                // whind shift angle varies between
                                                    // [-22.5 °,+22.5 °], parameters in sin function
                                                    // specify the wind shift velocity, here the wind
                                                    // shifts at a velocity of 2 %s

    begin_f_loop(f,thread)
    {
        F_PROFILE(f,thread,position)=5*cos(alpha*t*3.14/180);    // y component of airflow
                                                                    // varies due to variation
                                                                    // of alpha
    }
    end_f_loop(f,thread)
}

```

(c) Compile UDFs in Fluent

In menu bar select **Define >> User-Defined >> Functions >> Compiled**, a **Compiled UDFs** panel will appear as illustrated in Figure 4.1. In this panel press **Add** to choose a source file created in (b) and give a name in the **Library Name** field then press the **Build** button to compile the source UDF. The compiled file will be saved in a folder under the name given in the **Library Name** field. If the UDF is compiled successfully, Fluent will give some summary information as illustrated in Figure 4.2. Then in the **Compile UDFs** panel press the **Load** button to load the compiled UDF in Fluent and close the **Compile UDFs** panel.

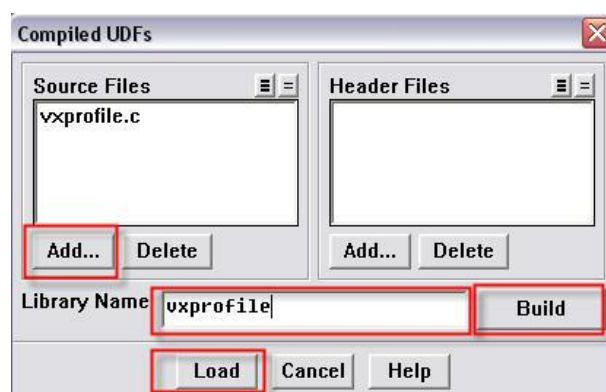


Figure 4.1 Compiled UDFs Panel

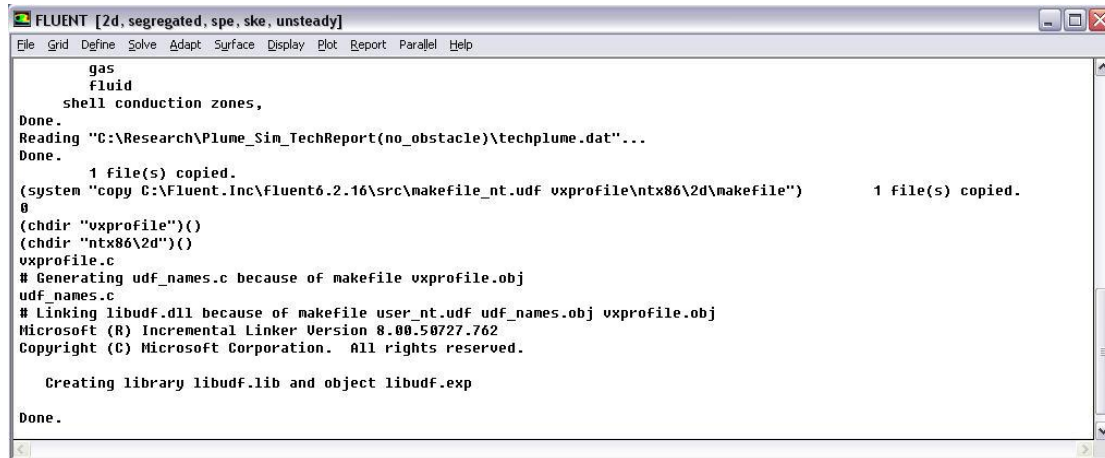


Figure 4.2 Compiled UDFs Summary Information

Notes: To successfully compile UDFs in Fluent, a C language compiler is needed. In this report, the C language compiler was provided by Microsoft Visual C++. Several environment variables needed to be set for VC++ before compiling UDFs. The environment variables are given as follows:

Variable	Value
include	root_path*\Microsoft Visual Studio 8\VC\include
lib	root_path*\Microsoft Visual Studio 8\VC\lib
path	root_path*\Microsoft Visual Studio 8\Common7\IDE;#
path	root_path*\Microsoft Visual Studio 8\VC\bin;#

* root_path is the location where MS VC++ was installed

the semicolon after the "IDE" is compulsory

4.3 Apply Customized Boundary Conditions Using UDFs

After the UDFs were compiled successfully, customized airflow inlet boundary conditions could be applied. As mentioned in Section 3.4, go to the **Boundary Conditions Panel** it can be found that there were two extra boundary conditions in both the x-velocity and y-velocity fields. The extra boundary conditions were defined by the UDFs and under the name specified in the UDFs. In the **Velocity Specification Methods** select **components** and in the **x-velocity** and **y-velocity** fields select the **inlet_x_velocity** and **intlet_y_velocity** respectively. Final settings of the customized boundary conditions are illustrated in Figure 4.3.

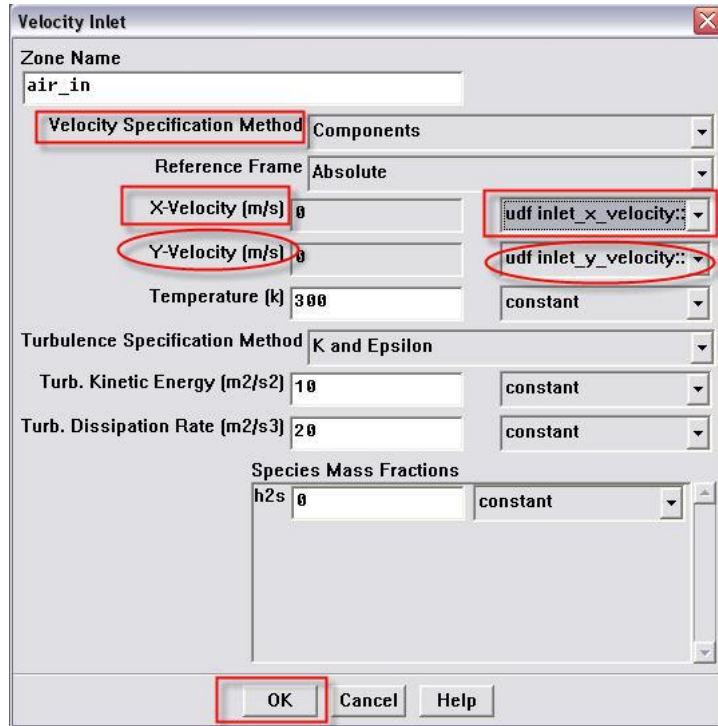


Figure 4.3 Final Settings of the Customized Boundary Conditions

4.4 Execute the Simulation and View the Simulation Results

Follow the steps given in Section 3.6, Section 3.7 and Section 3.8, specify the simulation time steps and view the simulation results. It can be seen that the plume was varied under the influence of wind shift. Simulation results are illustrated in Figure 4.4.

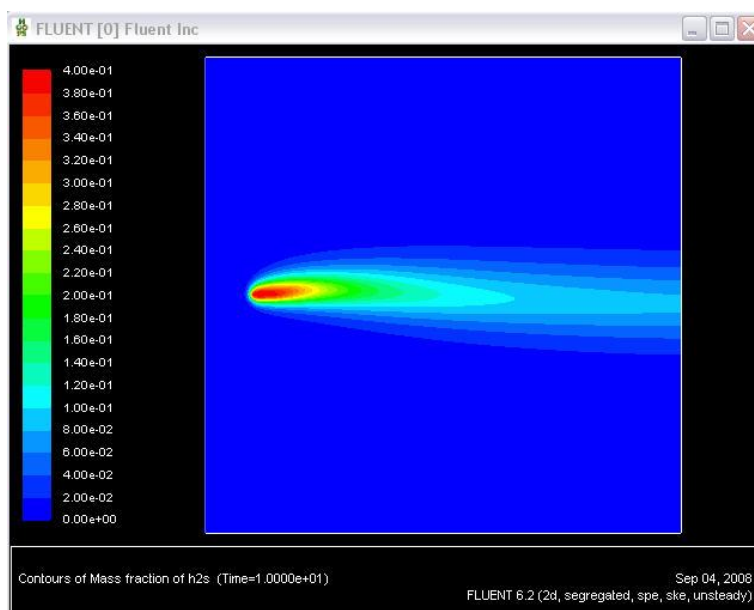


Figure 4.4 (a) Varying Plume Due to Wind Shift, T=10s

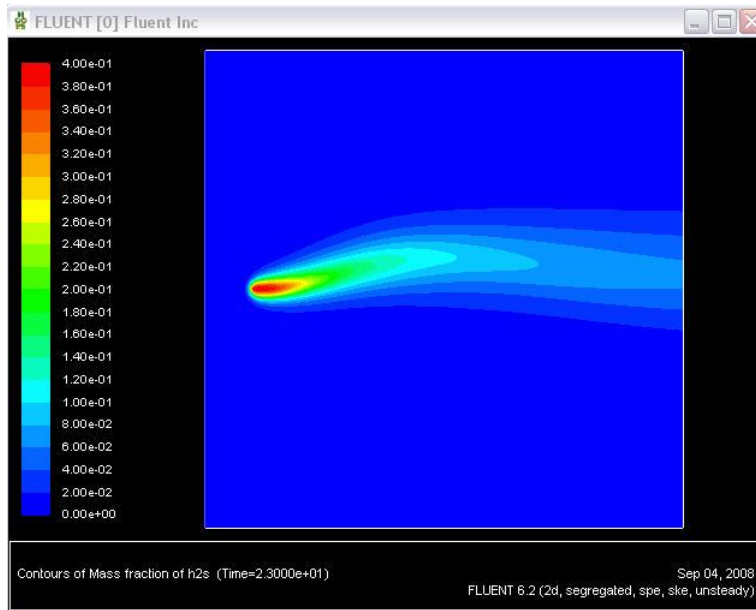


Figure 4.4 (b) Varying Plume Due to Wind Shift, T=23s

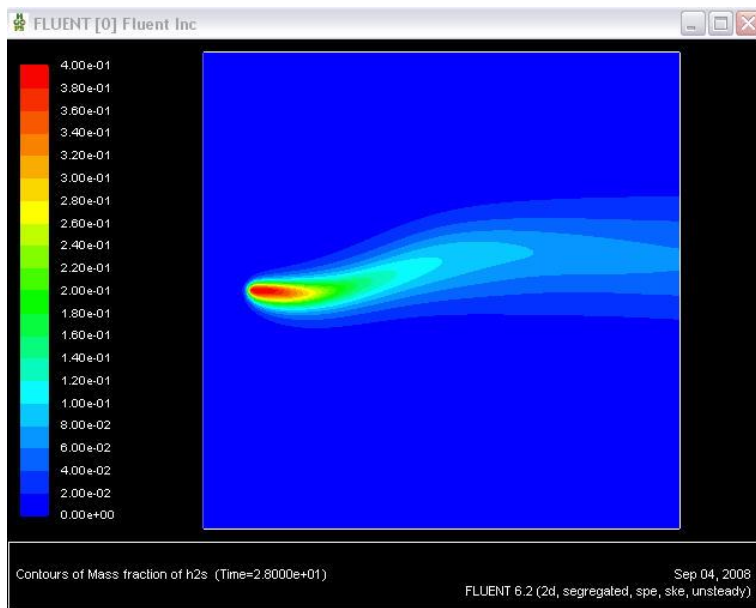


Figure 4.4 (c) Varying Plume Due to Wind Shift, T=28s

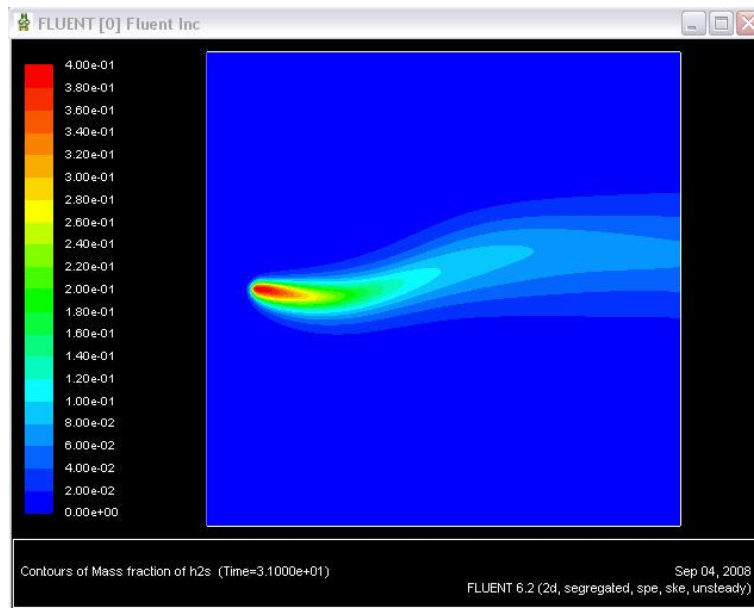


Figure 4.4 (d) Varying Plume Due to Wind Shift, T=31s

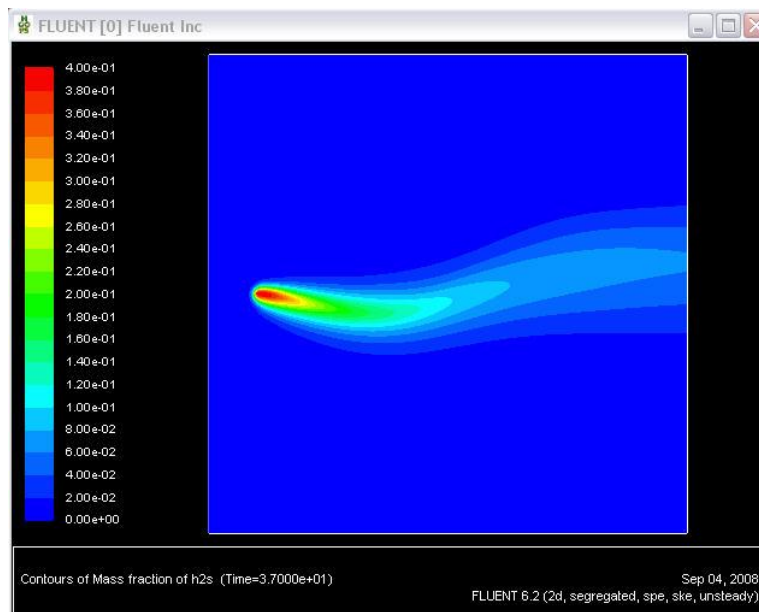


Figure 4.4 (d) Varying Plume Due to Wind Shift, T=37s

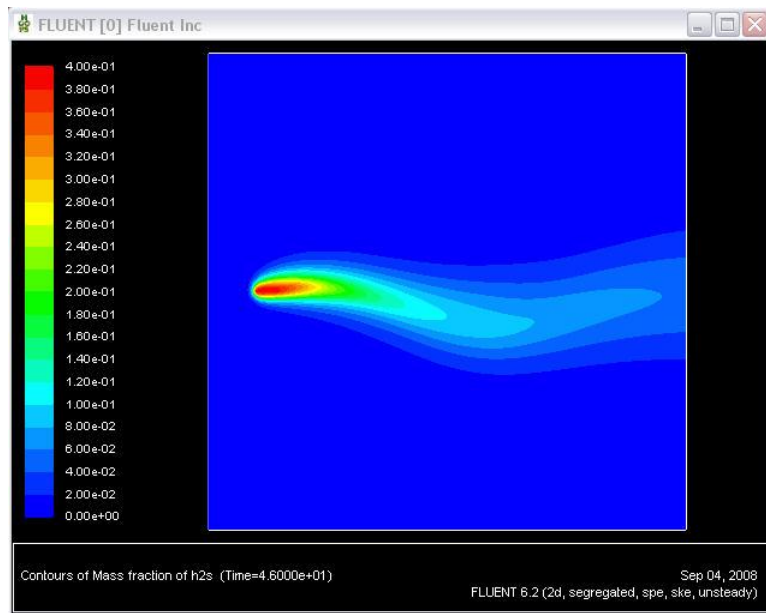


Figure 4.4 (e) Varying Plume Due to Wind Shift, T=46S

5 Simulation Data Post-Processing Using OpenDX

Fluent is one kind of CFD software good at calculation, however, it is not good at post-processing simulation data. For example, from the above sections, it can be seen that the odour/gas plume propagated during the simulation with very obvious concentration contours. However, in real world, it is almost impossible that we can see the plume concentration contours. Therefore, we need to adopt some approaches to visualize the simulation results which generated in Fluent to reflect a more realistic scenario. In this report, an open source package OpenDX was adopted to visualize the simulation data. Simulation data from Fluent were imported into OpenDX, OpenDX linearly mapped the plume concentration value to a colour map and presented concentration values with different colours.

5.1 Get Started

The first thing is to download the binary files for appropriate operation platform from www.opendx.org. In this report, the version for WindowsXP were downloaded. Next, install the software. View the README file included with the binary and it will give instructions on how to install and start up the executable. Once the OpenDX is completed installed, execute the program and the graphic interface of OpenDX will appear (Figure 5.1). Click the **New Visual Program** button, a **Visual Program Editor** window should appear on the screen and will look like Figure 5.2. A program which can visualize the Fluent simulation data will be constructed in the **Visual Program Editor**.

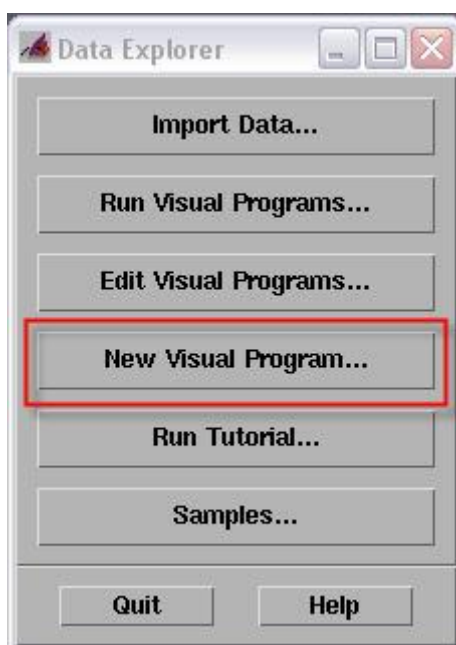


Figure 5.1 OpenDX Graphic Interface

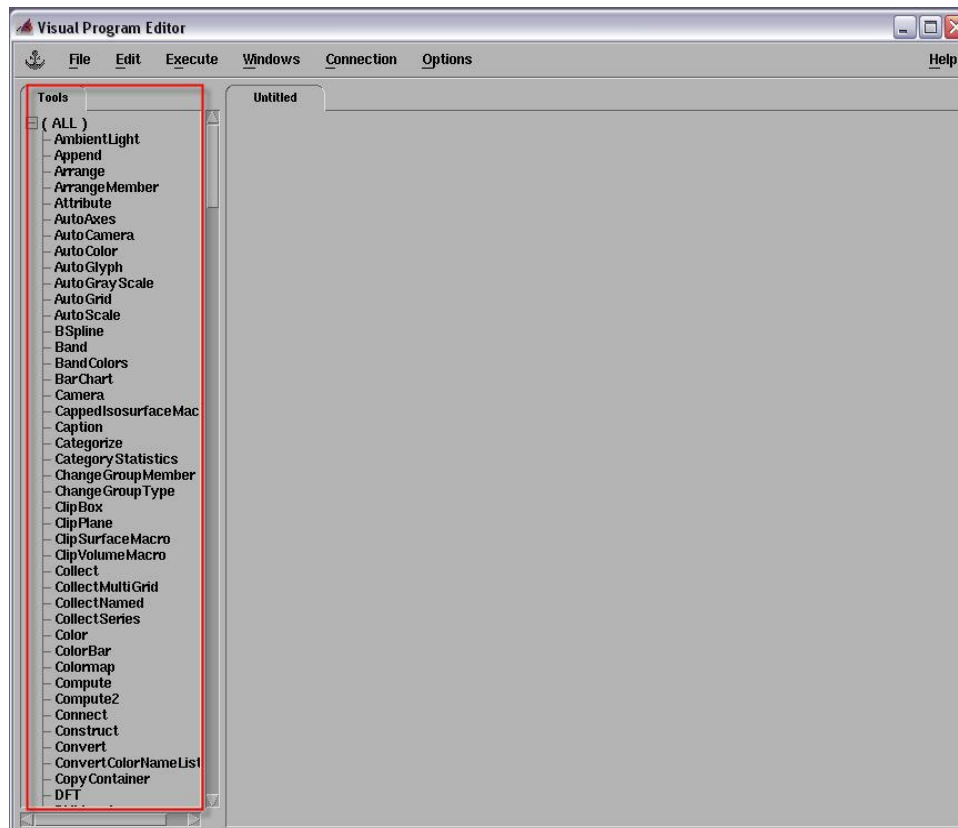


Figure 5.2 The Visual Program Editor Window

5.2 Construct a Data Visualization Program in OpenDX

In the left of the **Visual Program Editor** window, there are some predefined function modules which can be used to construct a program. Use the mouse to single click an appropriate predefined module and move the mouse to the right-hand side of the window. Position the mouse where you want to put the module and click the left mouse button once. A small green box should appear and with the name of the function module. The sample program used in this report is illustrated in Figure 5.3. Table 3 is a table demonstrates what modules should be selected in this report. "Tabs" are the little boxes on the top and bottom of the modules. The ones on the top represent input parameters, and the bottom ones represent outputs, which are used to connect to the inputs of other modules. If a tab is displayed outside of the module it is not set. If it is displayed inside the module than this parameter is already set. Connect the appropriate output tabs to the input tabs as illustrated in Figure 5.3. To accomplish this, click and hold the left mouse button on the output tab of the module. While still holding the left mouse button, drag the mouse to the input tab of another module. Now release the mouse, and a line should now be connecting the two modules.

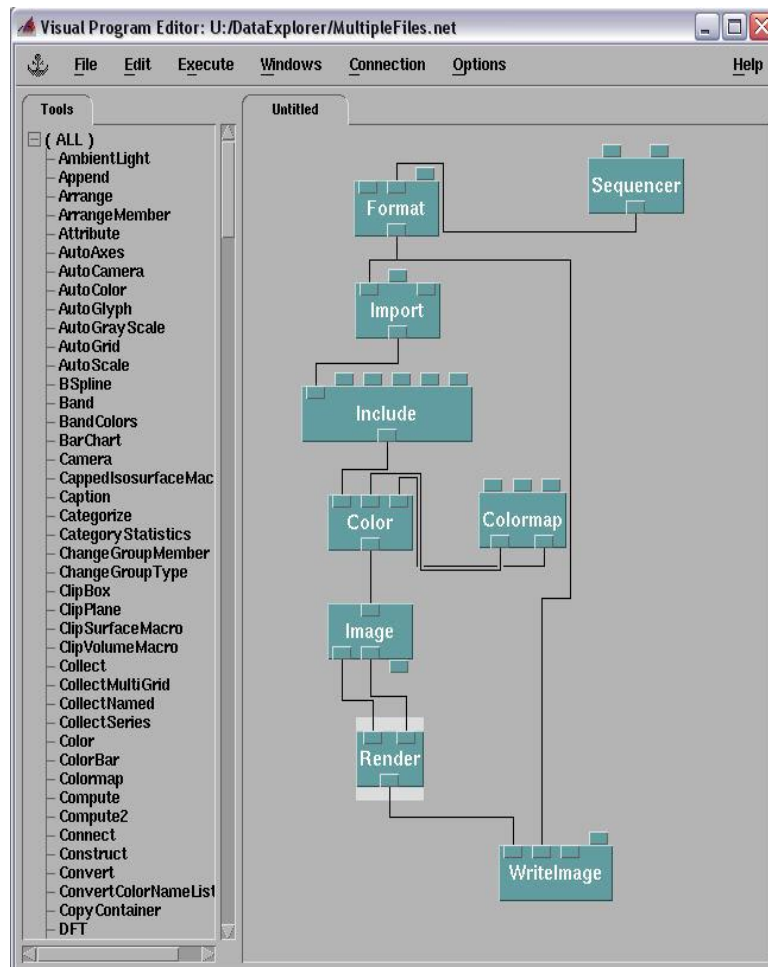


Figure 5.3 A Data Visualization Program in OpenDX

Table 3 Modules Adopted to Construct a Visualization Program

Module Name	Function
Format	Determine the data format which will be imported into OpenDX program. In this report, the data format is .dx.
Import	Import the data into OpenDX program.
Include	Include the data points in the data set. This module can be used to include a range of data which will be imported into the program.
Color	The color which represents the data.
Colormap	This module allows the users to create color maps that are applied to data.
Image	Display the data as an image.
Render	Render the image.
WriteImage	Save the rendered image as image files.
Sequencer	A loop function. When there are a sequence of data need to be visualized, this module can be used to imported and visualized the data one by one.

5.3 Parameter Settings for the Modules

After all the modules were connected, appropriate parameters for the modules need to be set. In this report, most of the modules adopted the provided the default parameters except the **Format**, **Import**, **Colormap** and **Sequencer** modules

5.3.1 Parameter Settings for the Format Module

Click left mouse button once on the **Format** module and press “Ctrl+F”. Set the parameters in the panel as illustrated in Figure 5.4. Press the **template** button under the **Name** field and input “%d” in the **Value** field. There is a sequence of simulation data from the Fluent simulation, therefore it is necessary to put the data in a loop and visualize the data using the same program. The setting here means the data which will be processed are under the name in sequence (e.g. 1.dx, 2.dx, 3.dx...). A **Sequencer** will provide the data names in sequence to this **Format** module.

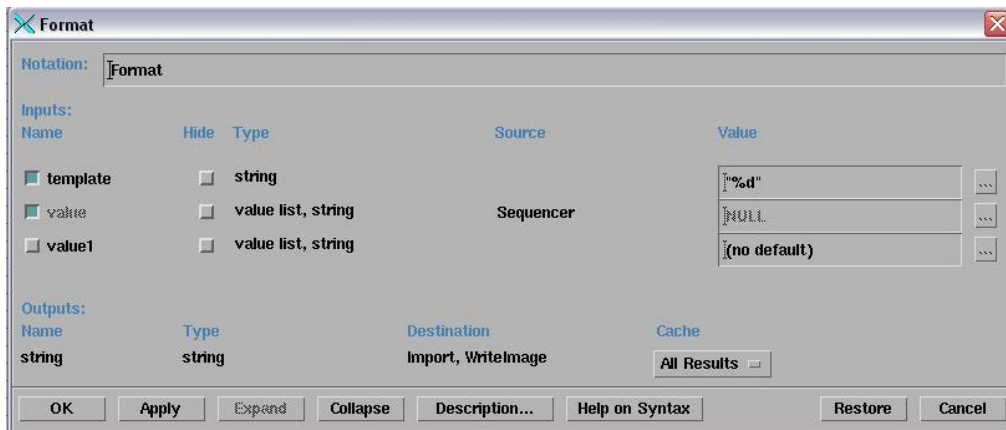


Figure 5.4 The Format Module Configuration Panel

5.3.2 Parameter Settings for the Import Module

Click left mouse button once on the **Import** module and press “Ctrl+F”. Set the parameters in the panel as illustrated in Figure 5.4. Press the **format** button under the **Name** field and input “dx” in the **Value** field. The setting here means that the data which imported in this program are in .dx file format which can be processed by OpenDX. Then press **OK** to close the panel

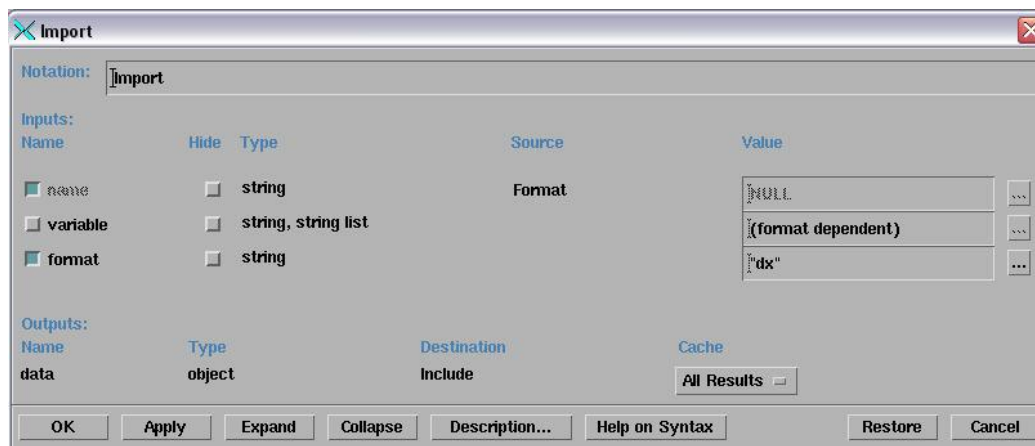


Figure 5.5 The Import Module Configuration Panel

5.3.3 Parameter Settings for the Colormap Module

Double click left mouse button on the **Colormap** module. A **Colormap Editor** should appear as illustrated in Figure 5.5. Input the maximum value and the minimum value in the highlighted area. The maximum value is the possible maximum value of the imported data and the minimum is the possible minimum value of the imported data. Adjust the **Hue** value by moving the control point until the colour used to represent the concentration value of the data is satisfactory then close the **Colormap Editor**. OpenDX will linearly map the concentration value of the imported data to the colour defined in the **Colormap Editor**. For instance, if the concentration value in the imported data is the highest, then the red colour will represent the highest concentration value.

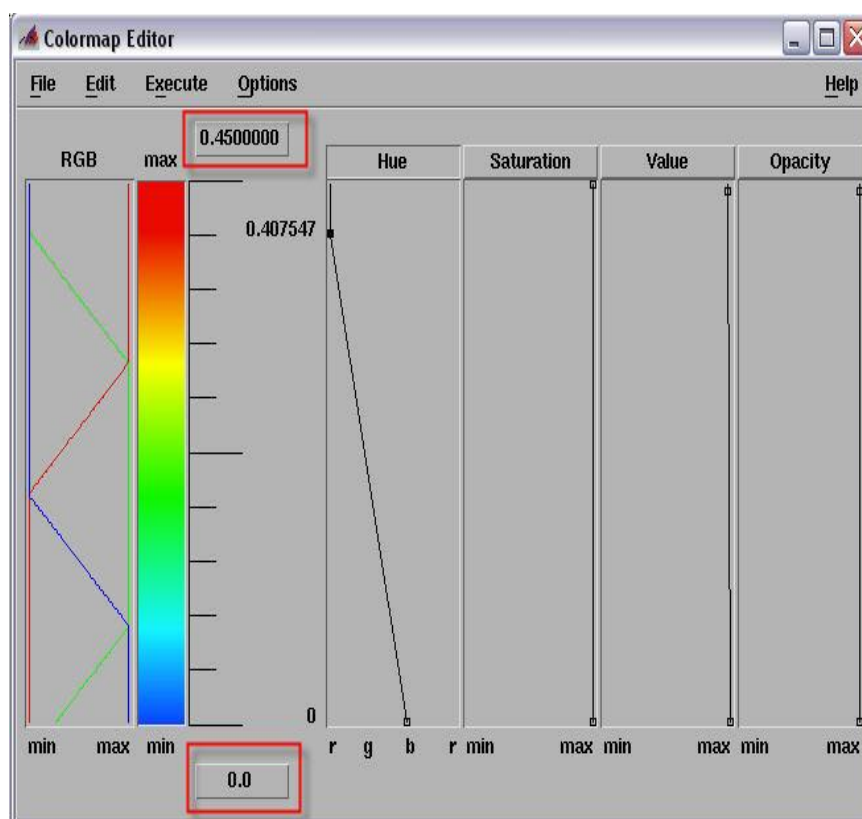


Figure 5.5 The Colormap Editor

5.3.4 Parameter Settings for the Sequencer Module

Click the left mouse button on the **Sequence** module once and press “Ctrl+F” hot keys. A Sequencer configuration panel should appear as illustrated in Figure 5.6. Press the **min** and **max** buttons in the **Name** field and input 1 and 200 respectively in the **Value** field. The settings here mean there are totally 200 data files in .dx format will be imported and processed using the program. These data file names begin from 1 and end at 200. The Sequencer module will provide the names of the files to the format module and the program will process all the imported data files one by one. Therefore, the **max** value in the **Value** field depends on how many data files need to be processed.

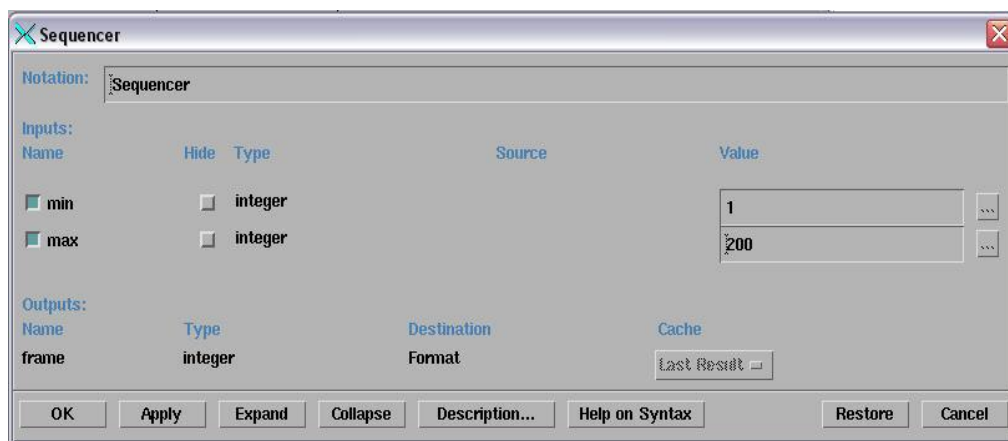


Figure 5.6 The Sequencer Configuration Panel

5.4 Execute the Data Visualization Program

After the parameter settings are done. In the menu bar, select **Execute >> Sequencer**, a **Sequencer Control** panel should appear as illustrated in Figure 5.7. Press the **play** button, the program will process all the simulation data from Fluent one by one. Figure 5.7 is an image of the visualized data using OpendDX



Figure 5.7 The Sequencer Control Panel

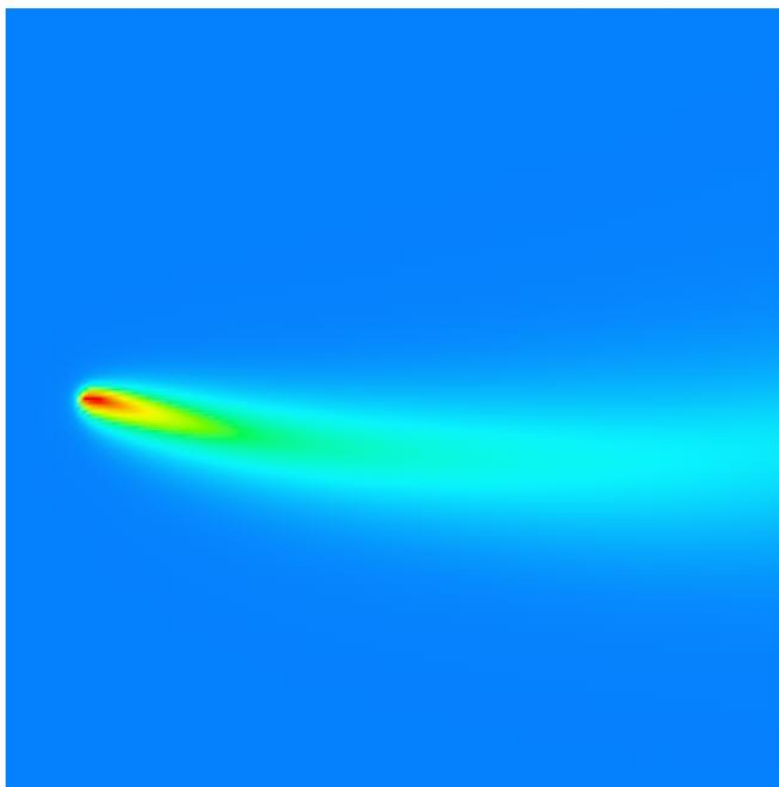


Figure 5.8 Snapshot of the Visualized Data Using OpenDX

Notes: To generate a sequence of simulation data in .dx file format, follow the steps given in Section 3.5 and enter the following commands in the **Command** field.

file/export/dx con%t.dx () h2s q

The following commands only export the h2s concentration value of the plume and the file names are:

data+time_step_number.dx

In this report, a sequencer was adopted to process the data files one by one, therefore it would be better to rename the data files in sequence, e.g. 1.dx, 2.dx, 3.dx...

Reference

[1] FLUENT 6.1 User's Guide, Fluent Inc.

[2] FLUENT 6.1 UDF Manual, Fluent Inc.

[3] OpenDX User's Guide,

OpenDX User's Reference.

IBM Visualization Data Explorer, website: <http://www.opendx.org/> last accessed

October, 2008.