LA-UR-07-2799

*Approved for public release;*
*distribution is unlimited.*

*Title:* QUIC Start Guide (v 4.5)

*Author(s):* Matthew Nelson, Bhagirath Addepalli, David Boswell and Michael Brown

*Intended for:* Public Distribution

Los Alamos National Laboratory, an affirmative action/equal opportunity employer, is operated by the Los Alamos National Security, LLC for the National Nuclear Security Administration of the U.S. Department of Energy under contract DE-AC52-06NA25396. By acceptance of this article, the publisher recognizes that the U.S. Government retains a nonexclusive, royalty-free license to publish or reproduce the published form of this contribution, or to allow others to do so, for U.S. Government purposes. Los Alamos National Laboratory requests that the publisher identify this article as work performed under the auspices of the U.S. Department of Energy. Los Alamos National Laboratory strongly supports academic freedom and a researcher's right to publish; as an institution, however, the Laboratory does not endorse the viewpoint of a publication or guarantee its technical correctness.

Form 836 (7/06)

# QUIC Start Guide (v 4.5)

## Quick Urban & Industrial Complex (QUIC) Dispersion Modeling System

Matthew Nelson, Bhagirath Addepalli, David Boswell and Michael Brown
Los Alamos National Laboratory

LA-UR-07-2799

last updated: 02/15/07

Unless otherwise indicated, this information has been authored by an employee or employees of the Los Alamos National Security, LLC. (LANS), operator of the Los Alamos National Laboratory under Contract No. DE-AC52-06NA25396 with the U.S. Department of Energy. The U.S. Government has rights to use, reproduce, and distribute this information. The public may \copy and use this information without charge, provided that this Notice and any statement of authorship are reproduced on all copies. Neither the Government nor LANS makes any warranty, express or implied, or assumes any liability or responsibility for the use of this information.

## Contents

Installation of the QUIC system ..... 1
What is QUIC? ..... 2
QUIC-URB ..... 2
QUIC-PLUME ..... 3
QUIC-PRESSURE ..... 4
QUIC-GUI ..... 5
  Starting the QUIC-GUI ..... 5
  Creating a new project ..... 6
  Opening existing projects ..... 8
QUIC URB ..... 12
City Builder ..... 13
  Add a New Building ..... 14
  Building height ..... 14
  Stacking Buildings ..... 17
  Adding a Pentagon Shaped Building ..... 20
  View Buildings in 3D ..... 21
  Editing Building Dimensions ..... 22
  Dimension Edit Box ..... 24
  Grid Resolution ..... 25
  Domain Size ..... 25
  Inner and Outer Grids ..... 26
  Geo-Referencing QUIC Domains ..... 28
  Vegetation ..... 31
  Add a Background Map ..... 32
  City Builder Options ..... 35
Met Generator ..... 39
  Logarithmic Profile ..... 40
  Power-Law Profile ..... 41

Urban Canopy Profile....42
Discrete Data Points (User-Specified) Profile....43
Import Data....45
Multiple Profiles....47
QUIC-URB Physics Options....51
Rooftop Algorithm....52
Street Canyon Vortex Algorithm....52
Upwind Cavity Algorithm....52
Street Intersection Algorithm....52
Convergence Criteria....52
Diffusion (Beta)....52
Running QUIC-URB....53
QUIC-PLUME....54
QUIC-PLUME Source Parameters....55
Agent Type....56
Source Type....56
Dense Gas....57
Distributed Particle Size....58
Explosive....63
ERAD....65
Multi-Source....66
Release Type....66
Source Strength....67
Source Geometry and Placement....68
Spherical Shell and Volume....69
Line....71
Cylindrical Area/Volume....72
Rectangular Area / Volume....73
Moving Point....74
Basic Agent Properties....75
Agent Library....77
QUIC-PLUME Simulation Parameters....78
View Current Collecting Box....80
Edit Collecting Box....81

Building Infiltration .....82
QUIC-PLUME Physics Options .....84
Non-Local Mixing.....84
Reciprocal Monin-Obukhov Length.....85
Boundary Layer Height.....85
Roughness Length.....85
Taylor Microscale (Beta).....85
Particle Splitting (Beta).....86
Running QUIC-PLUME .....87
QUIC Visualization GUI .....89
Show Legend .....92
Show Buildings.....93
Figure Toolbar .....94
Time and Position Controls .....95
Close All Plots .....95
Saving Animations.....95
Hold Plot .....96
Buildings / Ground.....97
QUIC-URB VISUALIZATIONS.....98
Vectors.....98
Contours.....110
Streamlines.....119
Pathlines.....121
Velocity Profiles .....126
QUIC PLUME VISUALIZATIONS.....130
Particles.....130
Contours.....137
3D Plume Visualization.....143
Surface Deposition.....147
QP Profile .....148
Plot controls .....150
QUIC Movie GUI.....151
Image Capture Modes .....152
Manual Mode.....152
Automatic Mode .....152
Flythrough Mode .....153
Automatic Mode with Flythrough .....154
Sequence Control .....155

DATA EXTRACTOR .....156
QUIC Data Extractor .....157
Appendix A.....168

# Installation of the QUIC system

- Unzip the QUICv4.4_ddmmyy.zip file into a single folder (a new folder named QUICv4.4 is suggested). Several subfolders, including a pcode folder containing the QUIC-GUI Matlab files and the QUIC-URB and QUIC-PLUME executables, will be created. You are now ready to run QUIC.

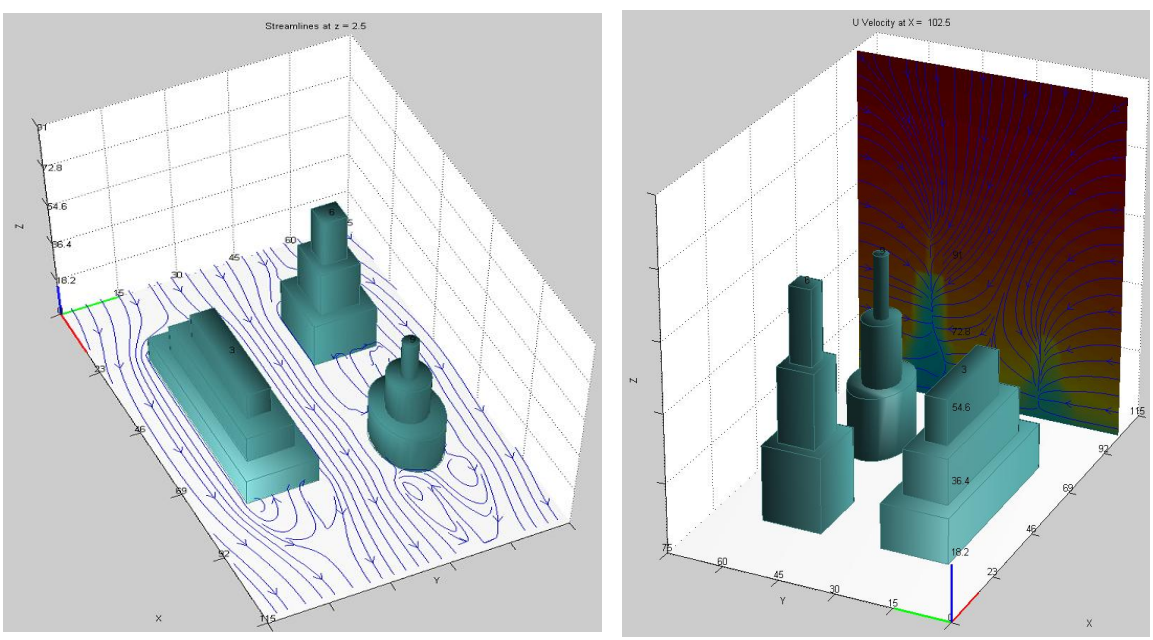
- In Appendix A you will find a complete listing of files that are shipped with the installation.

# What is QUIC?

QUIC stands for the Quick Urban & Industrial Complex (QUIC) Dispersion Modeling System. QUIC is a fast response urban dispersion model that runs on a laptop. QUIC is comprised of a 3D wind field model called QUIC-URB, a transport and dispersion model called QUIC-PLUME, and graphical user interface called QUIC-GUI. QUIC also includes a pressure solver, QUIC-PRESSURE, which solves for pressure fields in and around buildings using the velocity fields computed by QUIC-URB. CBR agent dispersion can be computed on building to neighborhood scales in tens of seconds to tens of minutes. QUIC will never give perfect answers, but it will account for the effects of buildings in an approximate way and provide more realism than non-building aware dispersion models.

## QUIC-URB

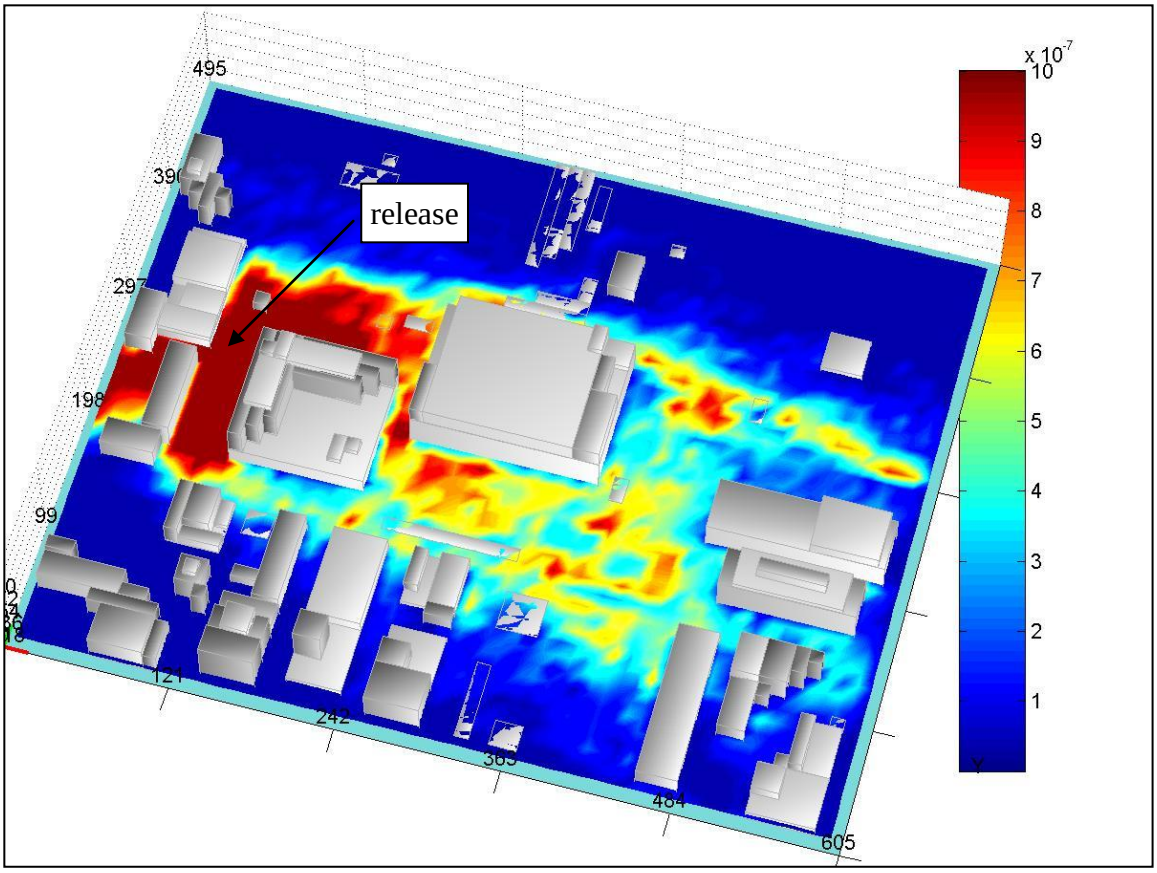
QUIC-URB is a fast response model for computing flow fields around buildings. It uses empirical algorithms and mass conservation to quickly compute 3D flow fields around building complexes. The underlying code is based on the work of Rockle (1990). Improvements to the original model are described in Bagal et al. (2003), Booth (2004), and Pol et al. (2006). Evaluation studies of QUIC-URB have been performed by Pardyjak & Brown (2002), Bowker et al. (2004), Singh et al. (2006), and Clark and Klein (2006). Below are examples of streamlines and contours of the flow fields calculated by QUIC-URB.




Left: Streamlines in the X-Y plane. Right: Wind speed contours and streamlines in the Y-Z plane in the downstream wake of buildings.

## QUIC-PLUME

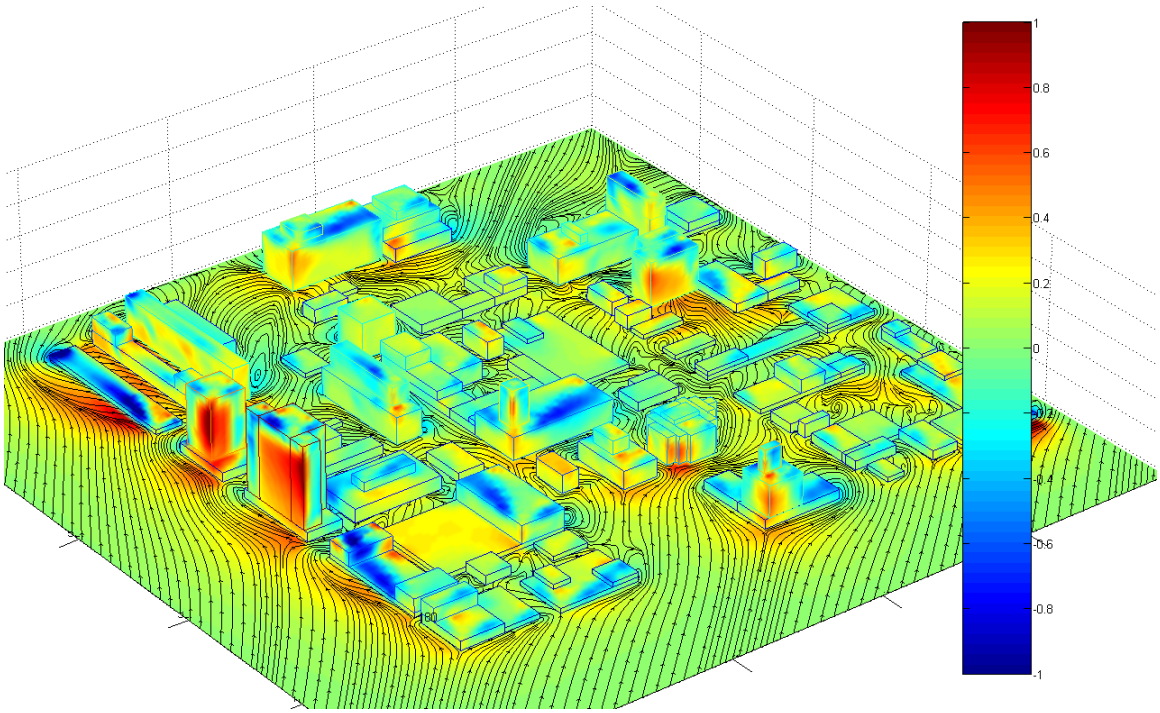
QUIC-PLUME is a Lagrangian random-walk dispersion model for computing concentration fields around buildings. It has been adapted to work in the inhomogeneous environment of cities. It includes more terms than the normal random-walk model in order to account for the 3D gradients in turbulent and mean flow fields. It includes reflection terms for building and street surfaces. The dispersion of aerosols and gases can be simulated, including deposition, gravitational settling and health properties. The code has been tested for both idealized and real-world cases (e.g., Gowardhan et al., 2006; Williams et al., 2004). For more information on the specifics of the code, see Williams et al. (2004).




Example output from the QUIC-PLUME model showing a contour plot of concentrations in the X-Y plane.

## QUIC-PRESSURE

Using the mean 3D velocity field produced by QUIC-URB, QUIC-PRESSURE solves for the 3D pressure fields through use of the pressure Poisson equation. Since only mean velocities are used in the computations, all turbulent contributions to the pressure field are neglected. More details on the pressure solver equations and evaluation studies can be found in Gowardhan et al (2006).




The image above shows the surface pressure coefficient for the Salt Lake City domain along with the near surface streamlines. The wind and pressure fields were created in 67 seconds and 45 seconds, respectively, using a 2.5 GHz Pentium 4 processor.

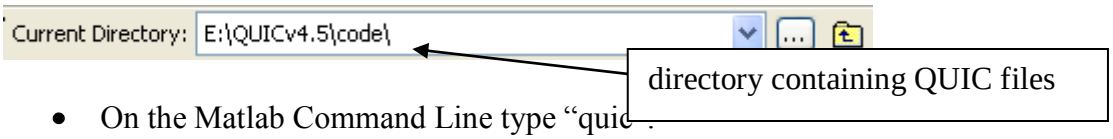
## QUIC-GUI

The QUIC-GUI allows one to set-up problems, run the wind and dispersion codes, and visualize results. The use of the QUIC-GUI is described below.

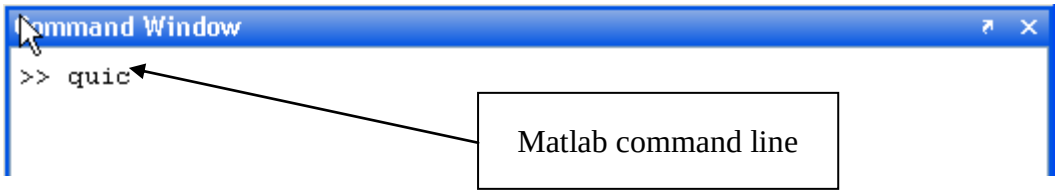
## Starting the QUIC-GUI

- Open Matlab

- Make the Matlab working directory the folder containing the QUIC files. Set the Current Directory to the folder named "code" or "pcode".

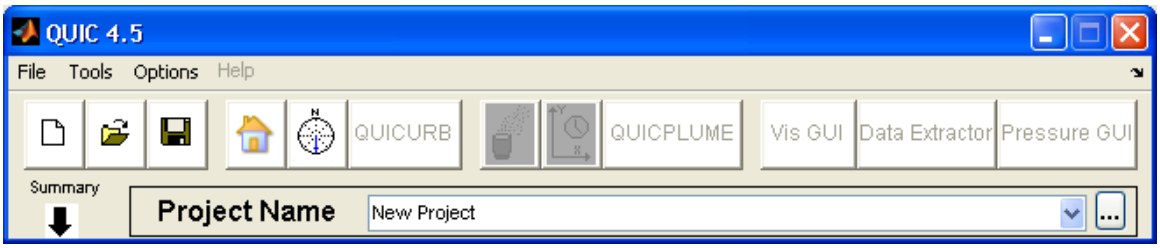



- On the Matlab Command Line type "quic".




- The QUIC-GUI Main window will open. This screen contains, from left to right, buttons for creating, opening and saving projects, specifying the input parameters for the QUIC-URB wind model (building layout and meteorological inflow), running QUIC-URB, specifying the input parameters for the QUIC-PLUME dispersion model (source term specification, concentration grid setup, and random-walk parameters), running QUIC-PLUME, visualizing the model output, exporting data, and computing pressure fields.

- The sequence of steps in setting up and running the QUIC models proceeds from left to right (City Builder > Met Builder > QUIC-URB > Source Term > Grid Setup > QUIC-Plume). Buttons are grayed-out (deactivated) until the prior steps are completed. The Vis-GUI becomes activated once output is produced by the QUIC-URB wind model.




Now a new project needs to be created or an existing project needs to be opened.

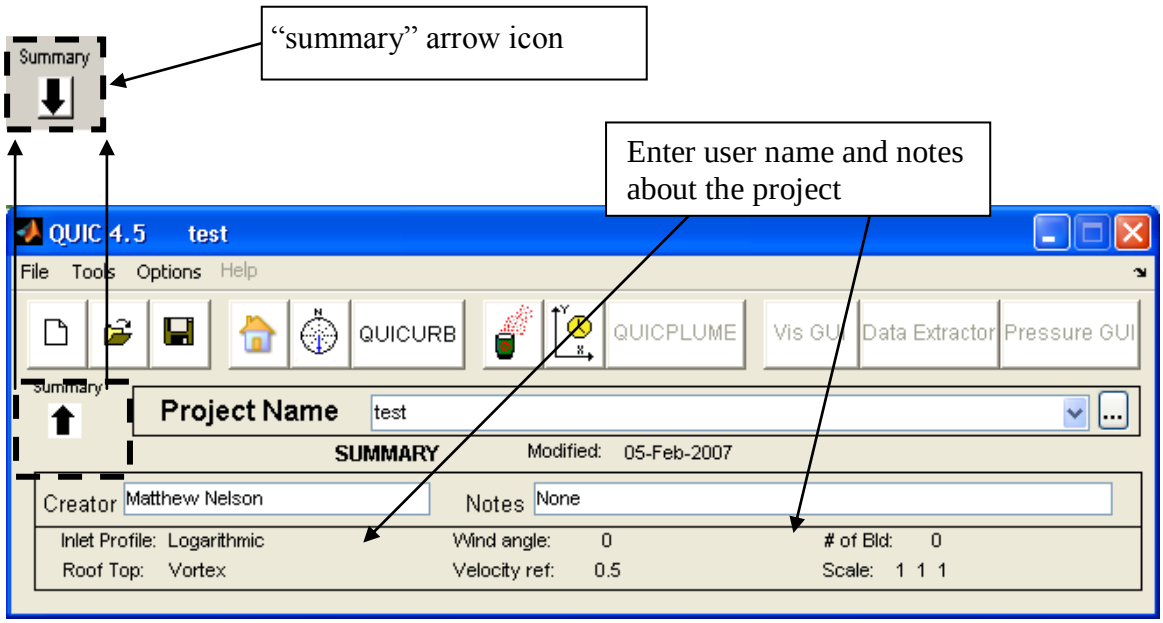
## Creating a new project

A new project can be created in one of two ways when first starting the software. One option is to enter the creator information and notes about the project (click the summary arrow to reveal the hidden project information panel) and click the save button, the other is to proceed directly to the City Builder and the user will be prompted to save the project upon exit from the City Builder. If the user is already working on a project, and wants to start a new project, the "new project" button can be clicked and the existing project will be removed from GUI memory and all values will be set to their default value (note that you can lose the existing project, unless the user has saved the project). Saving a project using the "Save" option is shown below.

**Optional Step:**

Press the summary arrow icon

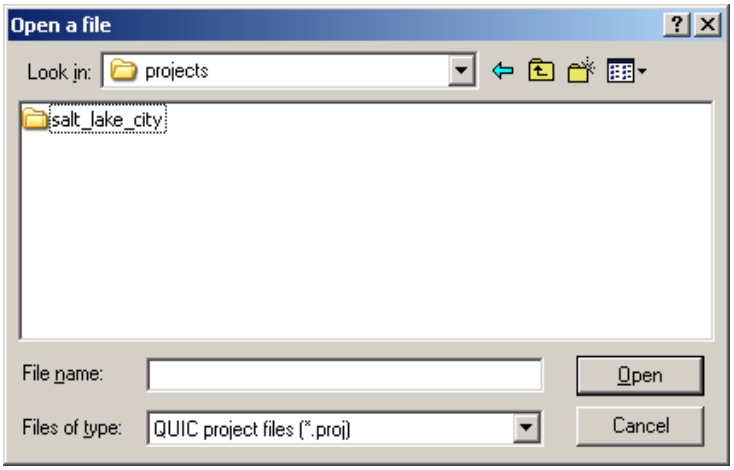
- Enter information into the creator and notes text fields (optional)



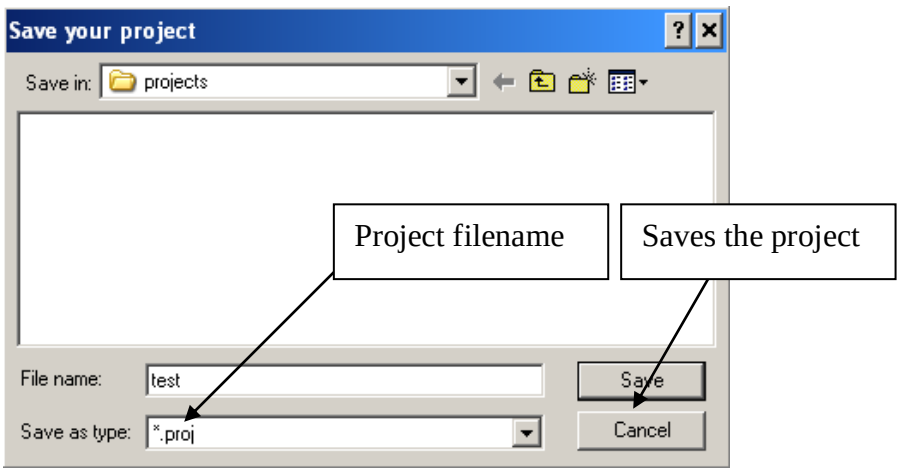

- To save the new project click and a pop-up window will open.

- In the pop-up window select the folder into which you want to save your project. A convenient choice is the \Projects folder found in the \QUICv4.4 directory.

However, project files can be saved in any folder on your computer. Below is an example folder structure with the project folder containing a folder that has the files for a specific project.




- Type in the file name (note: this name will also be used to name your project folder). Then press 'Save'. A project folder will be created which contains project files (*.proj).




- The last directory where a project was saved is written into the "default_options.info" file located in the code/pcode folder and the GUI will use this location as the default projects folder the next time that it is opened.

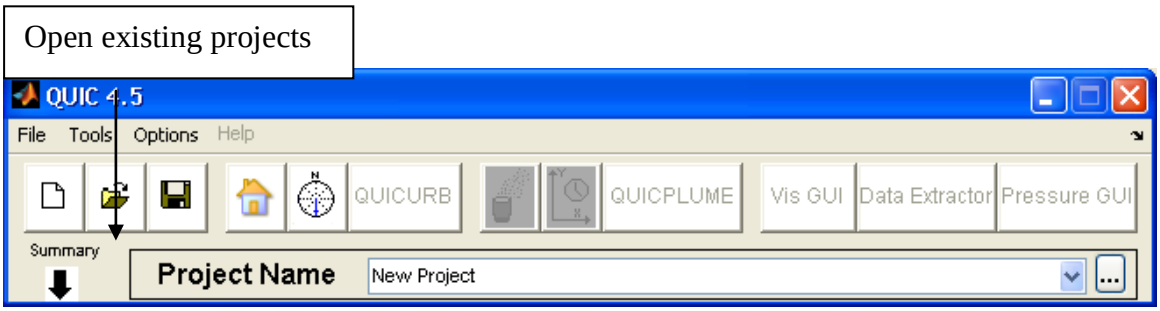
Note: If a project is not created at this time, the GUI will prompt the user to create the project before exiting the City-Builder.

## Opening existing projects

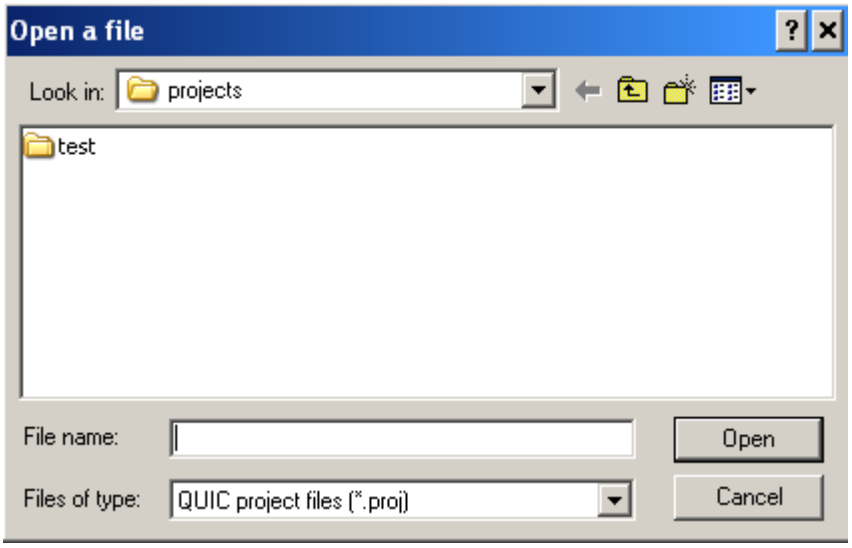
There are two options for opening existing projects: the first using the open project button, and the second using the Project Name pull down menu. The browse icon is used to navigate between different project directories (folders where multiple projects are stored).

### Open Project

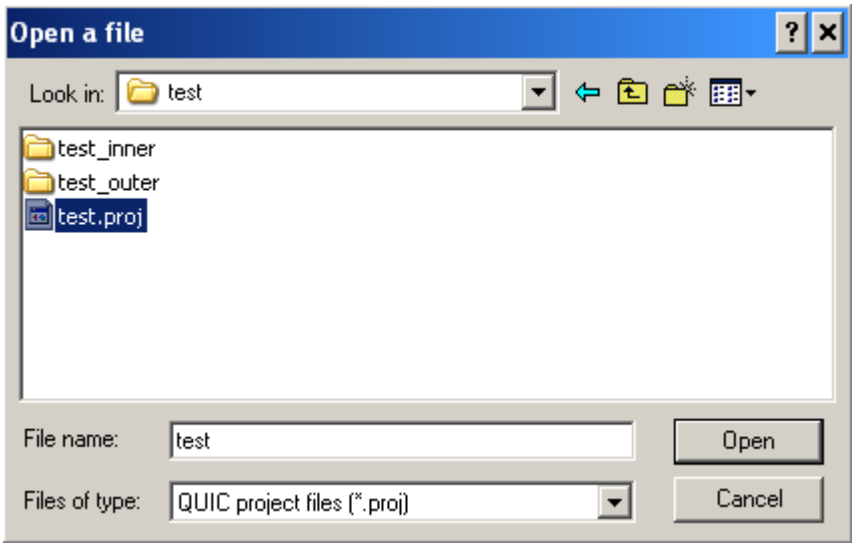
To open an existing project using the open project button, press the button




Pressing the 'open' button opens a browse window. Navigate to the /Projects folder or wherever you have saved the QUIC project of interest.

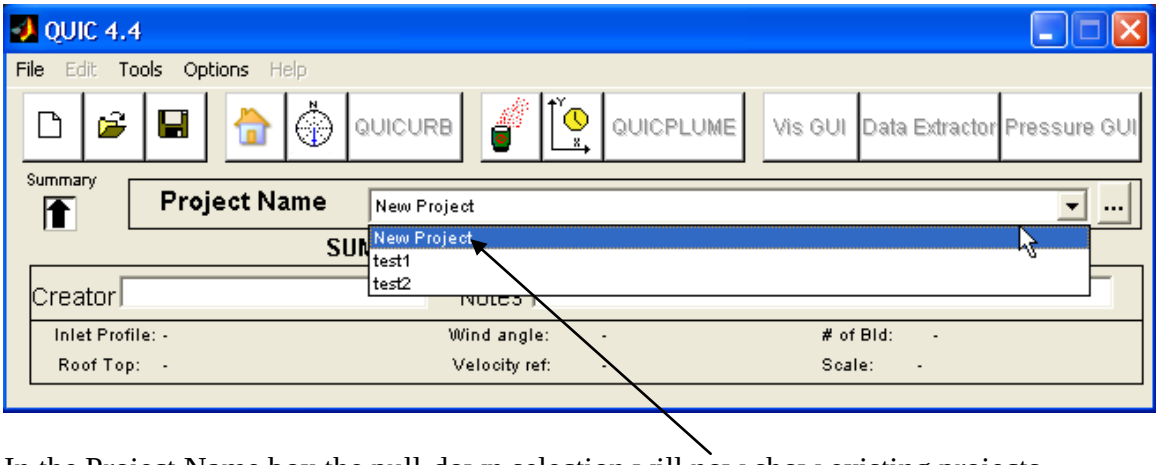


Double-click on the project folder of interest, and then select the *.proj file. This project will now be loaded into QUIC-GUI. This is show below.




**Project Name Pull-Down Menu**

Existing projects can also be opened using the Project-Pull-down menu



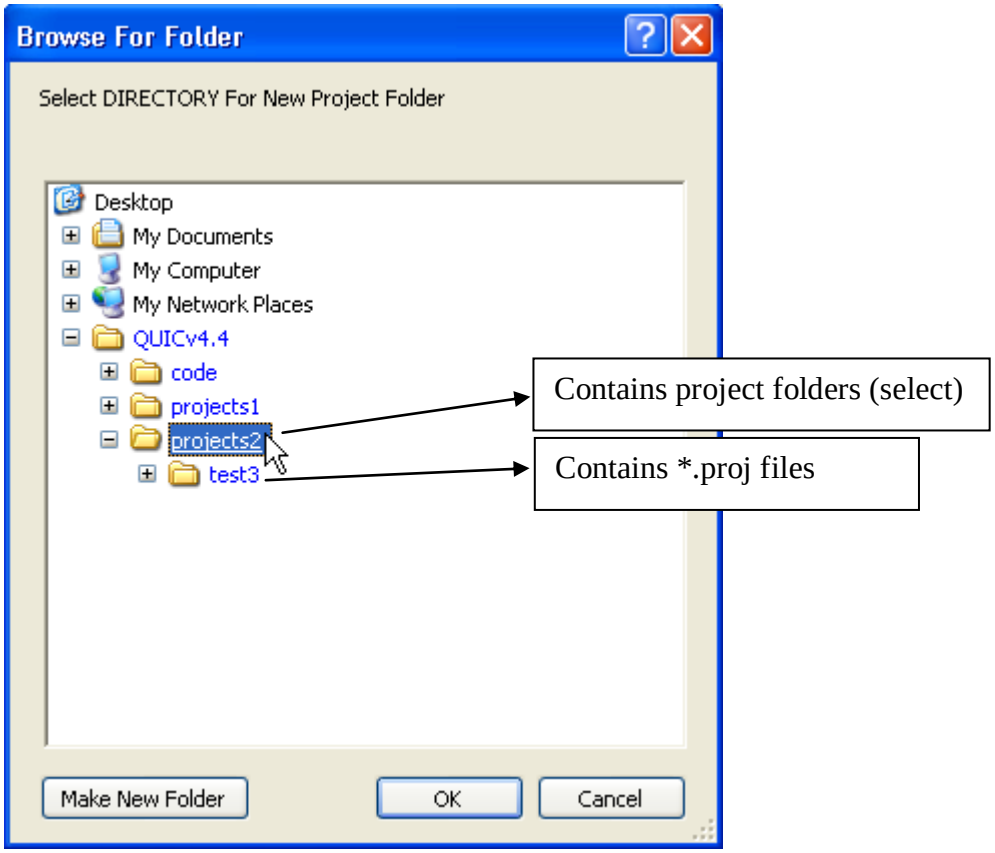

In the Project Name box the pull-down selection will now show existing projects

Note: the *.proj and associated input files can be modified outside of the QUIC-GUI as long as the parameters are chosen properly and the format is kept the same.

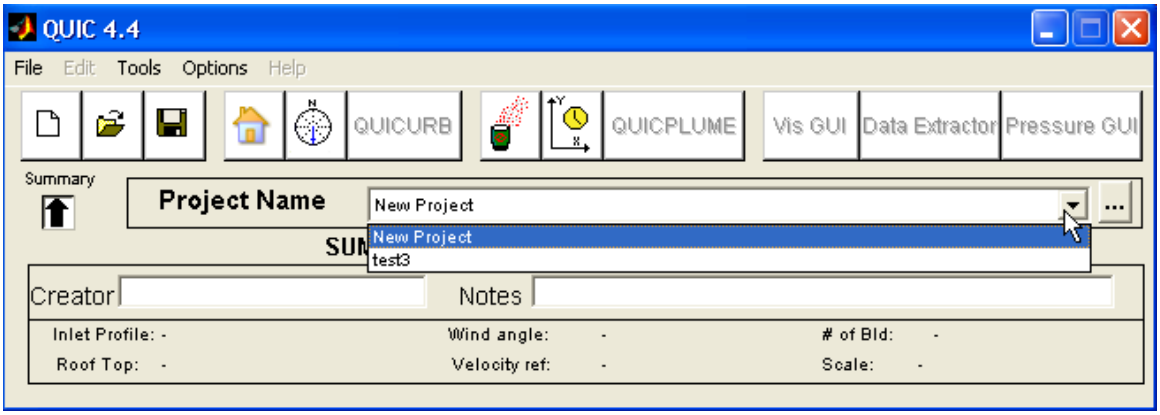
**Browse Icon** ...

To populate the project pull down menu with projects saved in a different project directory press the button ...

Pressing this button launches the 'Browse For Folder' pop up window




Left-click on the project directory of interest and press the OK button at the bottom of the GUI. This will populate the Project Name Pull-Down Menu with the projects found in the new Projects directory as shown below.




The projects found in the newly selected projects directory can now be loaded from the Project Name Pull-Down Menu as was described above.

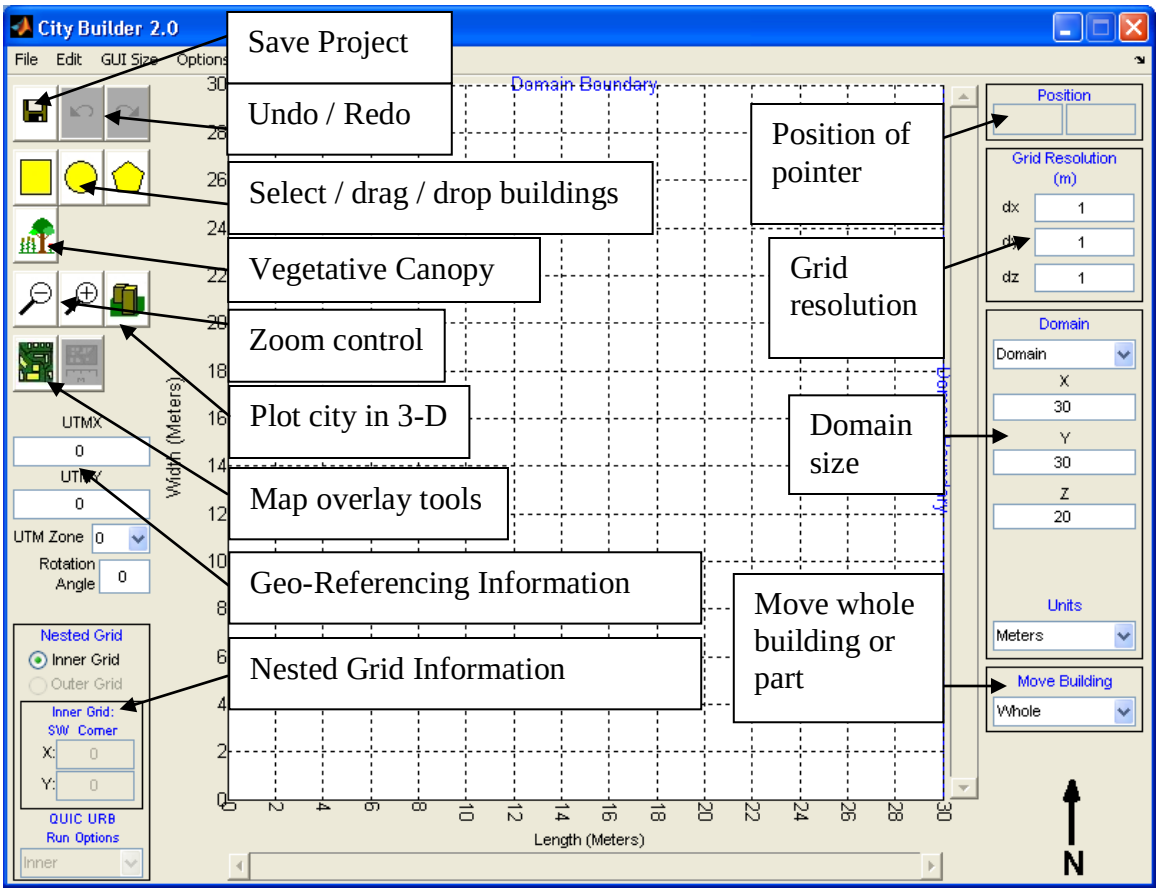
Once the appropriate project is opened, one can then proceed with setting up the parameters needed to run the QUIC-URB wind code. The first step in this process, adding or modifying buildings in the City Builder, is described in the next section.

# QUIC URB

# City Builder

- To open City Builder click on the house icon on the main QUIC-GUI pop-up window.

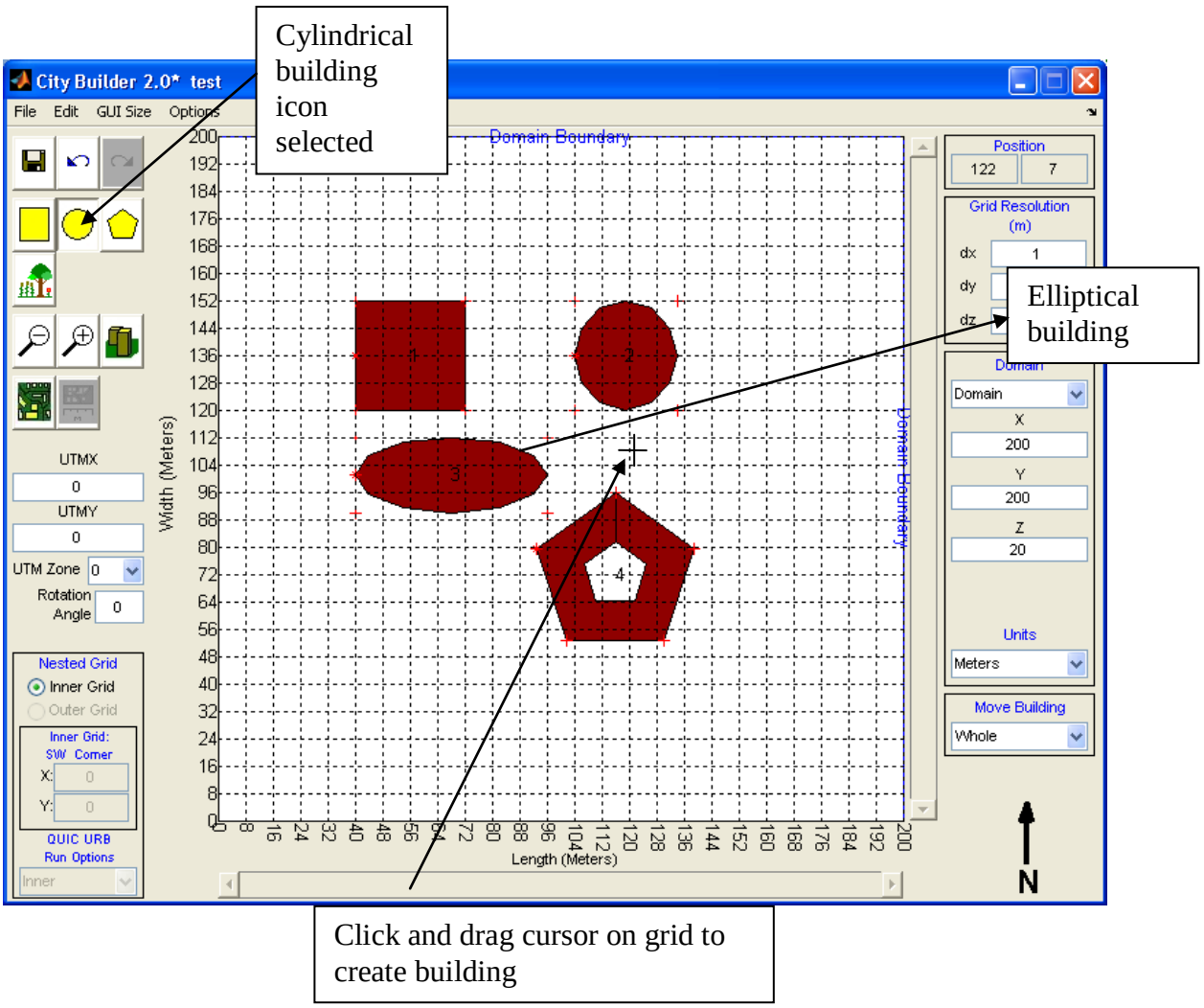
A building layout interface will open similar to the one shown below.



## Add a New Building

To add a new rectangular or cylindrical building to the project: select the building type; Left-click on the grid at the location where the building is desired, drag the building footprint out to the desired size, and release the left mouse button. A special pentagon-shaped building is also available.

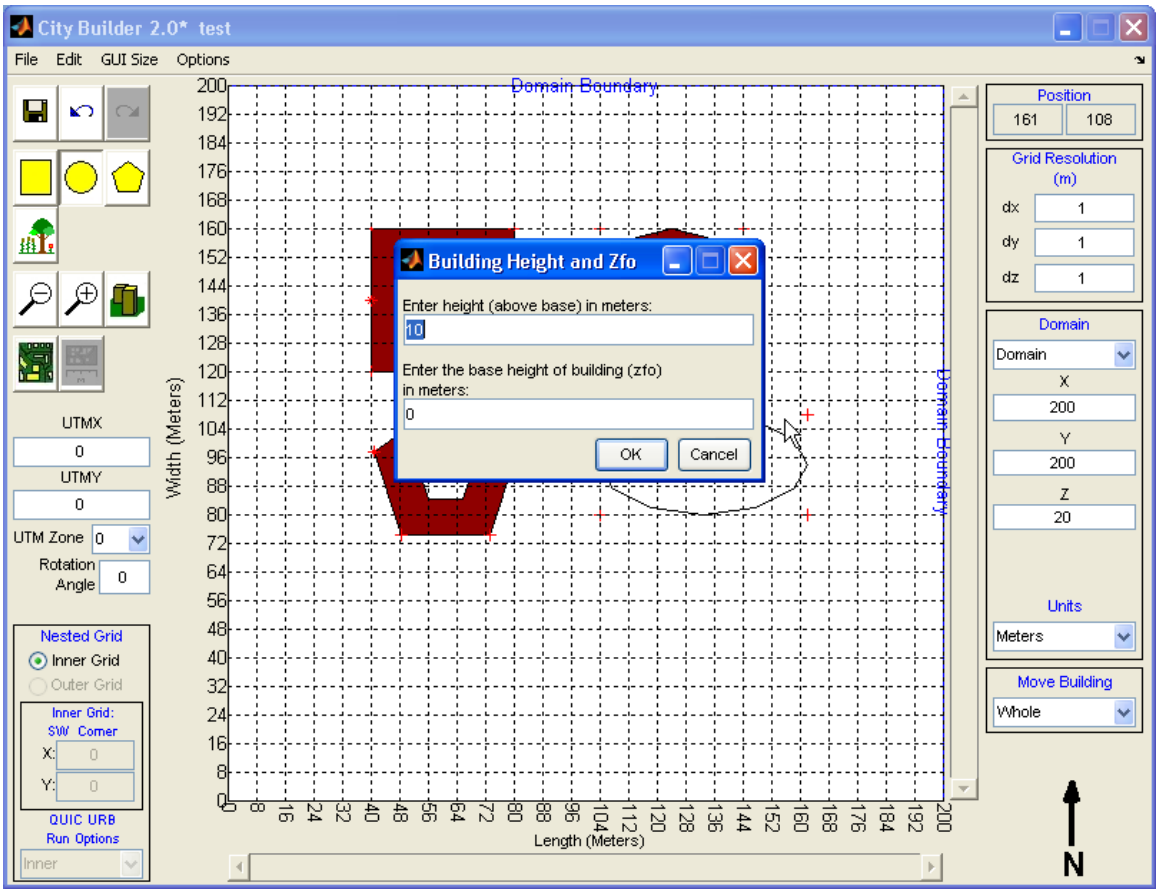
Note: Dragging out an oblong footprint with the cylindrical building selected produces an elliptical building.



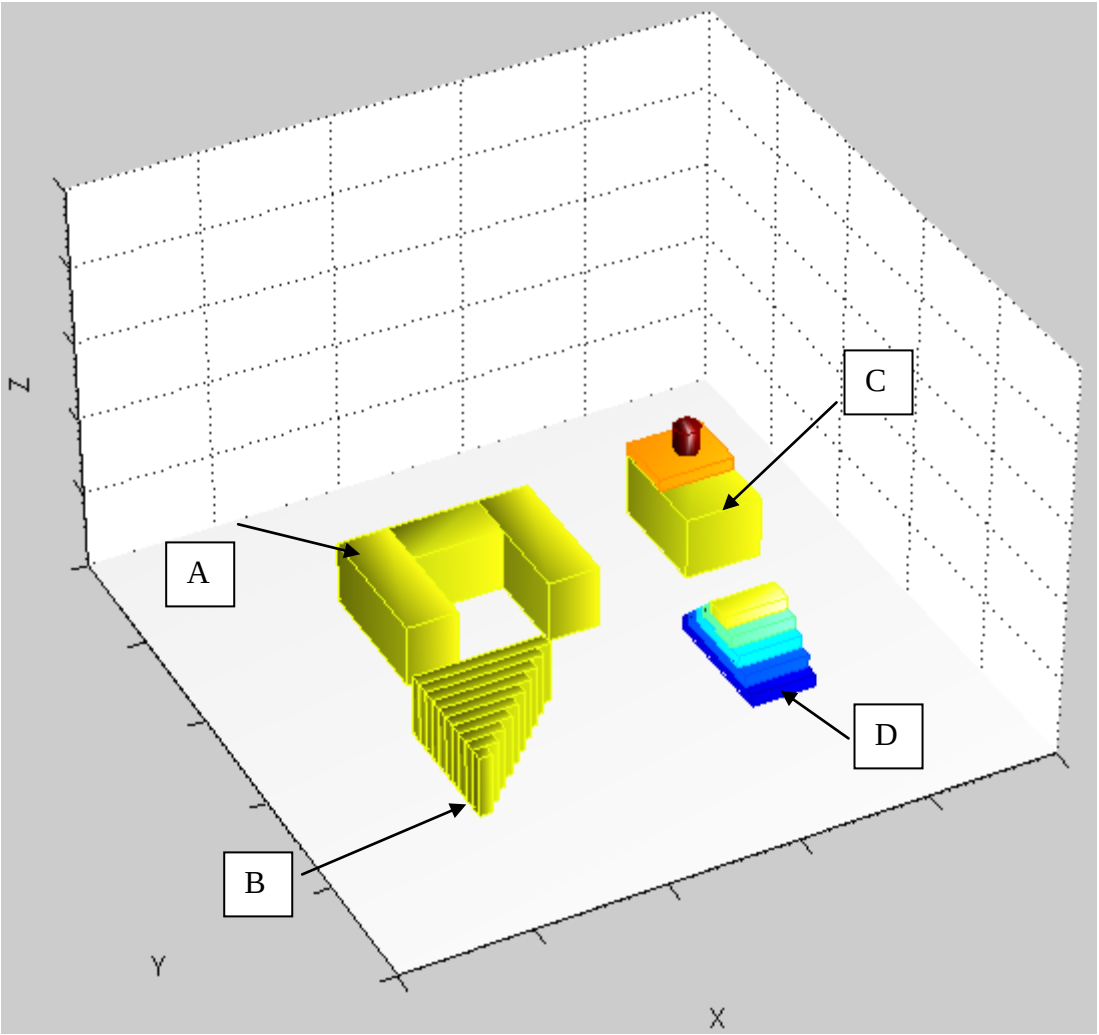

## Building height

A pop-up window for building height appears after dragging the building footprint.

Enter desired height and click OK. Note that the base height of the building is used (i.e., is non-zero) when stacking buildings (see next section).




Complex buildings are made by placing multiple basic building shapes together either side-by-side and/or stacked. In this manner, L-shaped, U-shaped, and non-orthogonal (stair-stepped) buildings can be created as shown below.

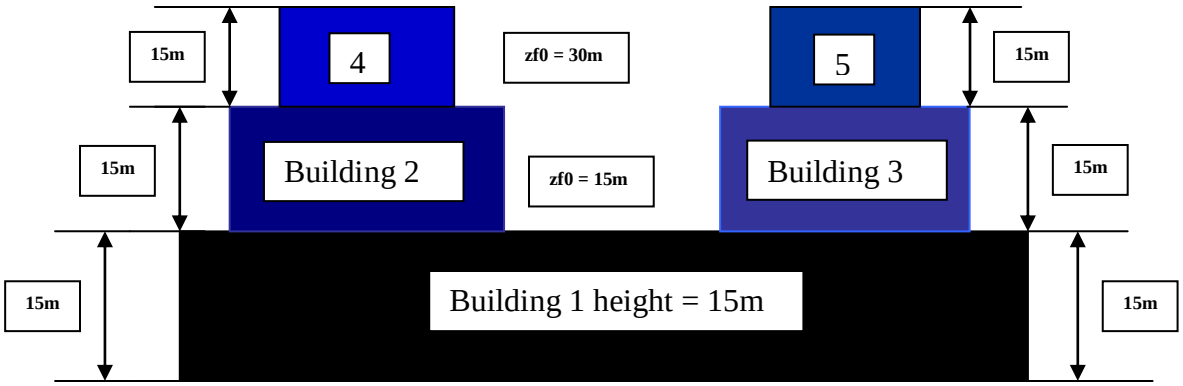


Complex buildings shown above are constructed using multiple basic building shapes. The buildings shown above are colored by height. Building A has an open courtyard giving it a U-shape. Building B is an approximation of a building with a triangular footprint. Building C has a section of the roof that is higher than the rest with a cylindrical smokestack protruding from the higher portion of the roof. Building D has a pitched roof.

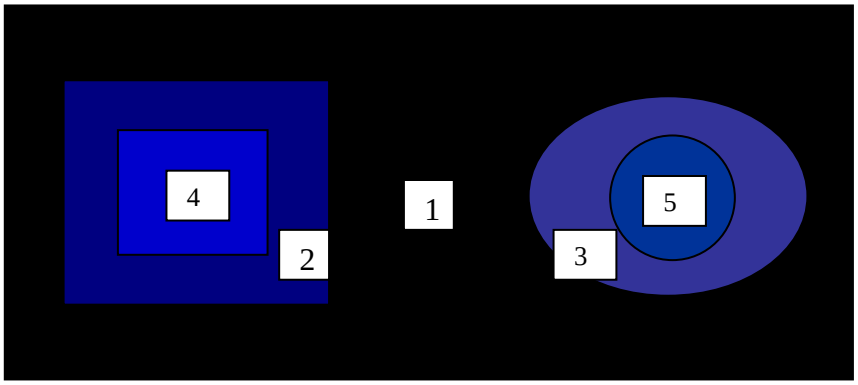
## Stacking Buildings

When a building is added by dragging out a shape, a pop-up window appears for setting the building height. QUIC-GUI has the ability to "stack" building shapes. 'zfo' is the base height of the building. This new parameter is needed to specify whether the building block is resting on the ground surface (zfo=0) or on another building block (zfo is then the height of the underlying building). The zfo of a new building will automatically be set to the height of the building below it if the starting point of the dragging is over another building.

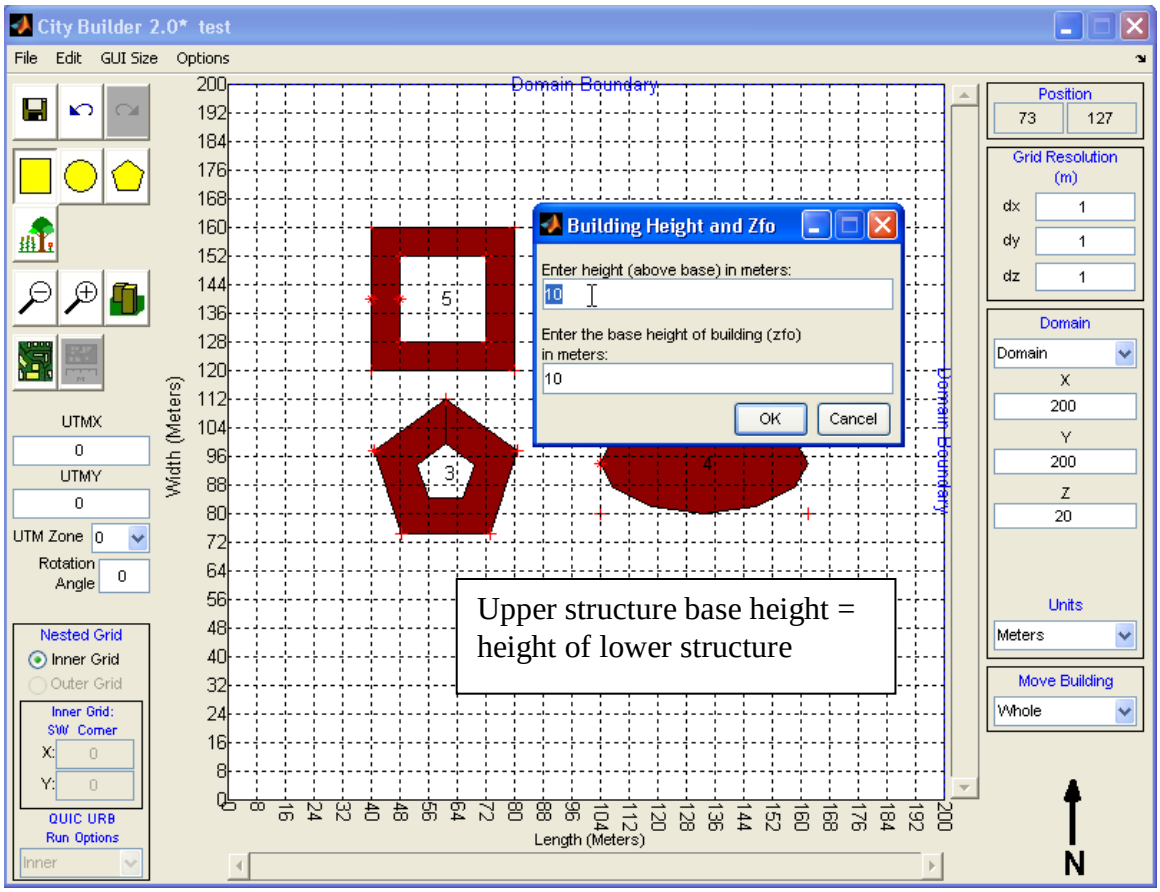
In the following example the lower building (Building 1) has a height of 15 meters. Therefore, zf0 values for buildings 2 and 3 are entered as 15m. The heights of building's 2 and 3 are set to 15m. Hence, zf0 for buildings 4 and 5 are entered as 30m. Note that the height of the building is the height of the actual building block, not from the ground. So 15 m should be entered for the height of building 4, not 45 m.



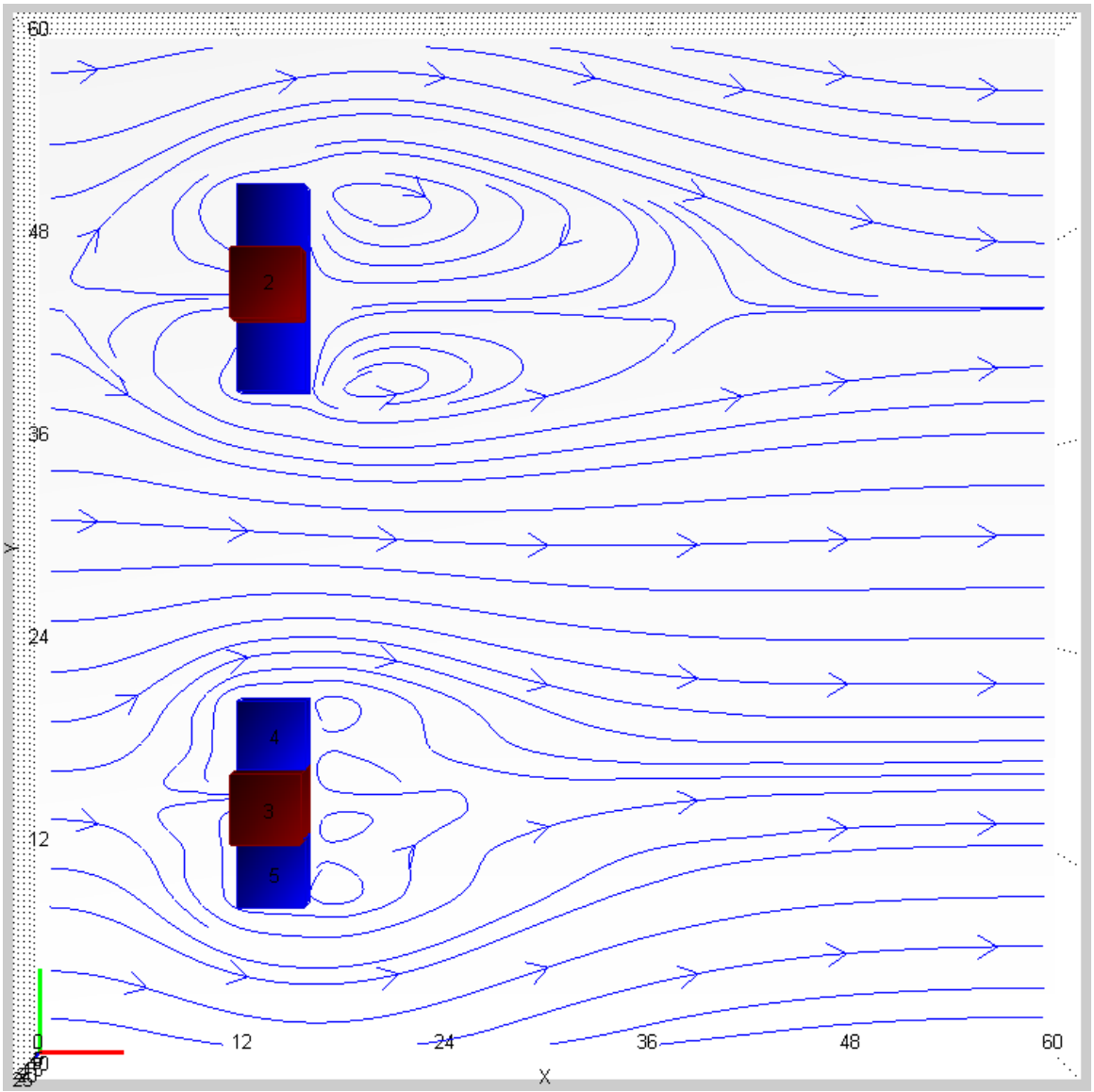

**Front View**




**Top View**



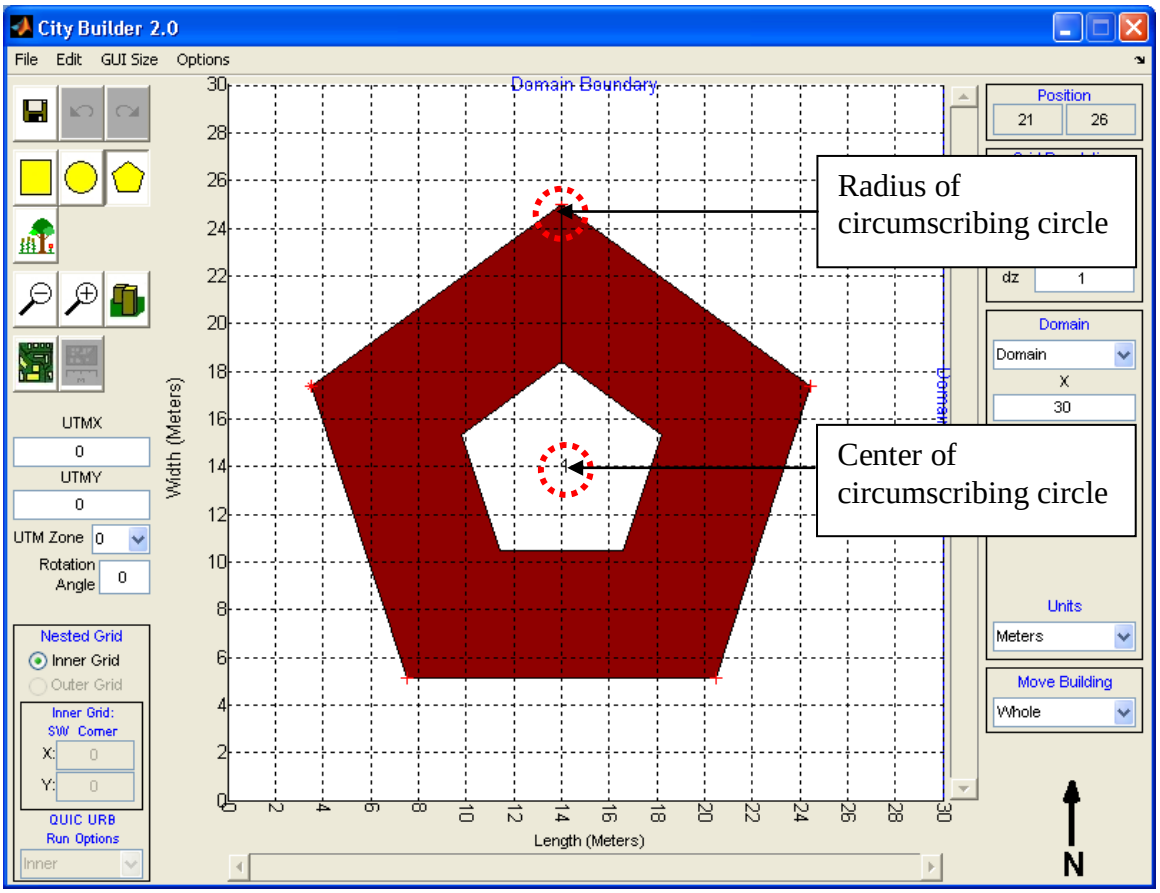

Note: Complicated building shapes may be constructed by either stacking several simple building shapes or placing them side-by-side. The QUIC-URB model has been developed to provide better results from stacked buildings. Whenever possible, buildings should be stacked, rather than placed side by side. Building structures that are geometrically identical, but built by stacking vs. side-by-side can produce different wind fields. This is shown in the figure below.




The graphic above shows an X-Y plane of streamlines around the base of two geometrically identical buildings. The building at the top of the image was created by stacking one building on top of another. The building at the bottom was created by placing three buildings side-by-side. An orthogonal view of these two buildings is found below in the View Buildings in 3D section.

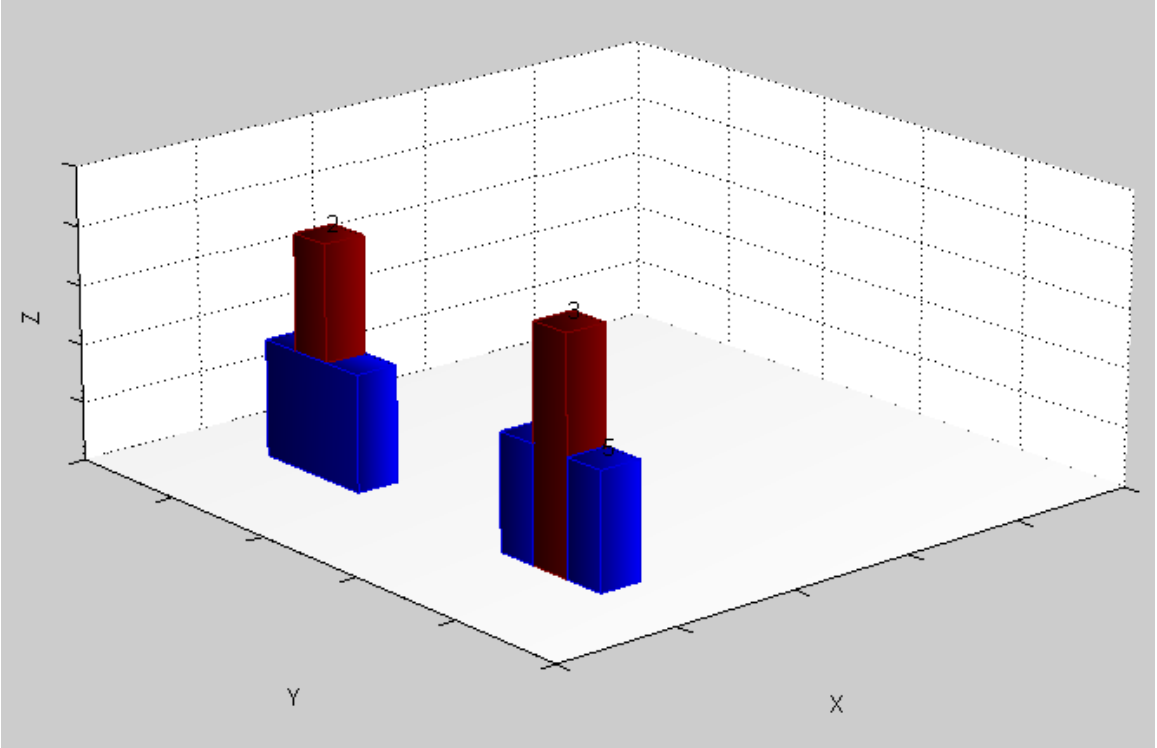
## Adding a Pentagon Shaped Building

The method for dragging out a pentagonal building is different from that used to define rectangular and cylindrical buildings. A circumscribing circle defines the pentagonal building. As such a pentagon is produced by left-clicking on the center of the circle and dragging out the radius of the circumscribing circle. Due to the highly specialized nature of the pentagon shaped building the flow algorithms in QUIC-URB have been tuned for the 12:1 radius to height ratio characteristic of the Pentagon building. The flow fields produced from these algorithms may not be appropriate for radius to height ratios that are dramatically different from 12:1.



## View Buildings in 3D

The 3D Plot button brings up a 3D viewer window, which allows the user to look at the buildings in 3D from different angles.

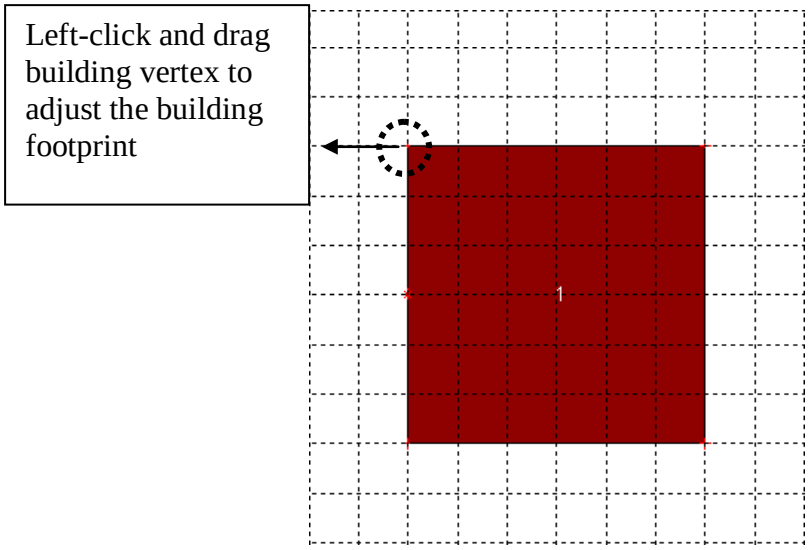



This is the orthogonal view of the example above involving geometrically identical buildings that are constructed by stacking two buildings or placing three buildings side-by-side.

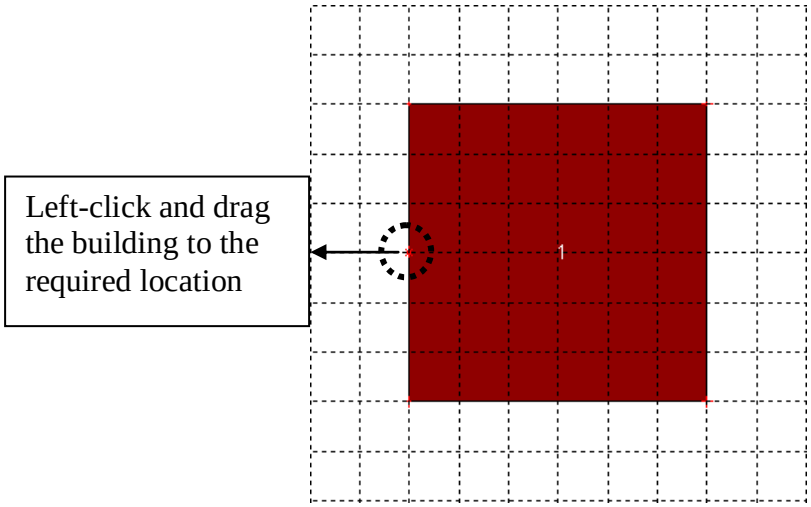
Once the buildings have been added, the Save button must be clicked in order to continue on with the QUIC-URB simulation. Alternatively, when the City Builder is closed, a pop-up window will request that the user save changes.

## Editing Building Dimensions

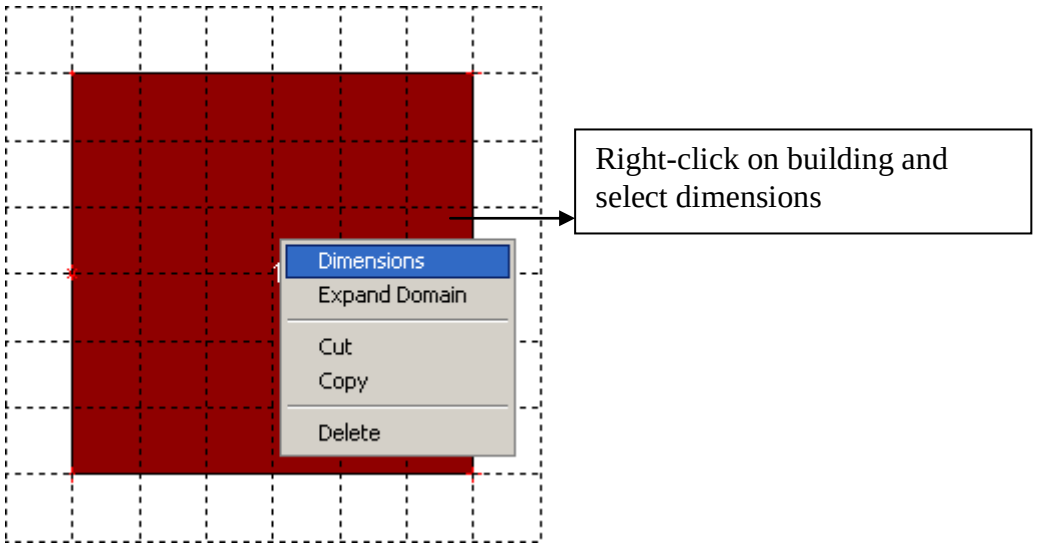
Dimensions of existing buildings can be adjusted by left-clicking one of the red vertices that define the outer extent of the building and dragging it to the desired building footprint. The building dimensions will automatically snap to the raster grid used by QUIC. This is shown in the figure below.

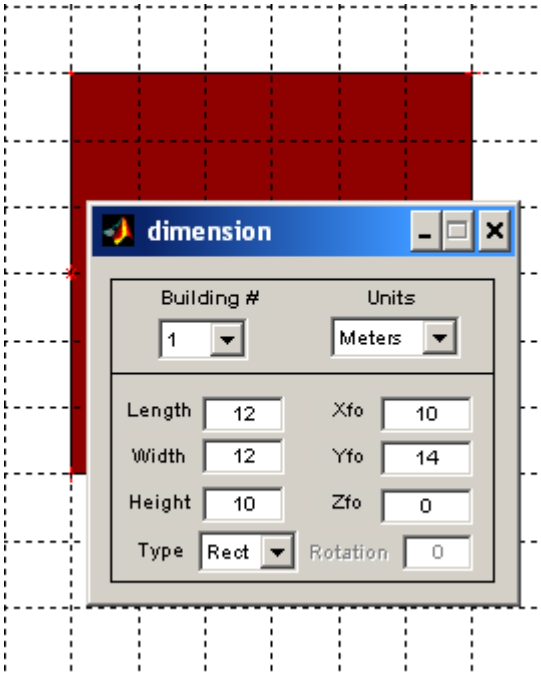



Buildings may be moved horizontally and vertically by left-clicking the red star that defines xfo and yfo for that building (the middle of the left-hand side of the building) and dragging it to the desired location on the grid. The building location will automatically snap to the raster grid used by QUIC. This is shown in the figure below.

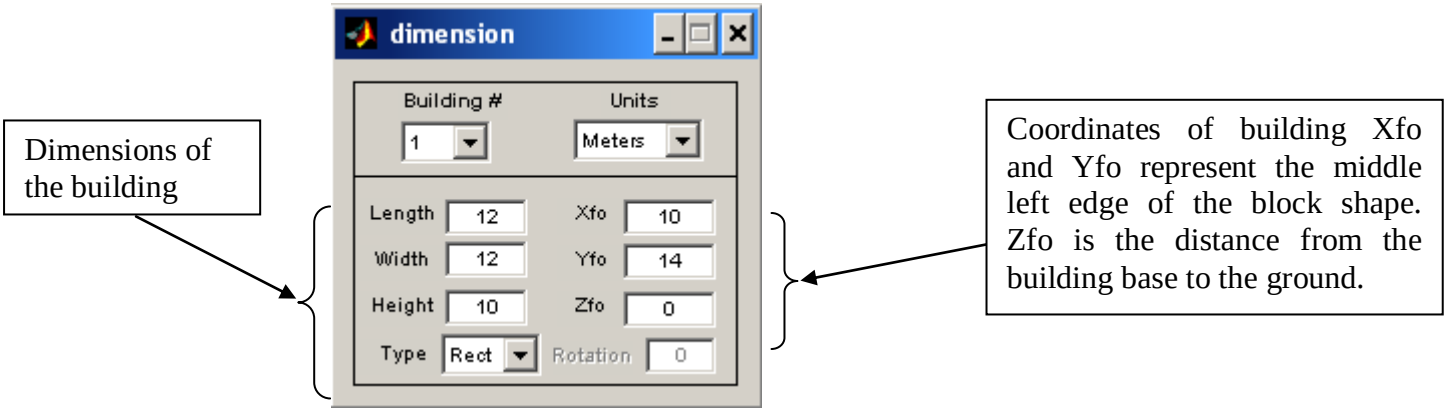


Alternatively, attributes of existing buildings can be edited by right-clicking the building and selecting “dimensions”. This launches the Dimension Edit Box, where all building attributes may be adjusted. Building height, base height (zfo), and rotation angle (only available for pentagonal buildings) can only be adjusted using the Dimension Edit Box. This is shown below.



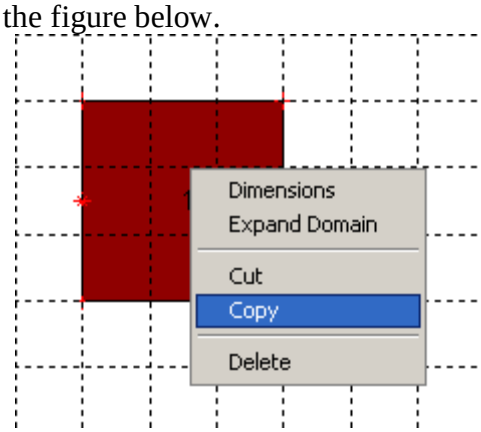



## Dimension Edit Box




## Copy Building

To copy a building, right click on the building and select 'copy'. Then, at any other point in the domain, right click and select 'paste' to copy buildings from one location to another. This is shown in the figure below.



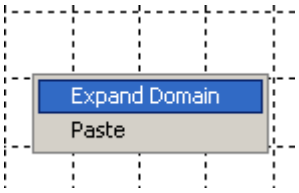
## Grid Resolution

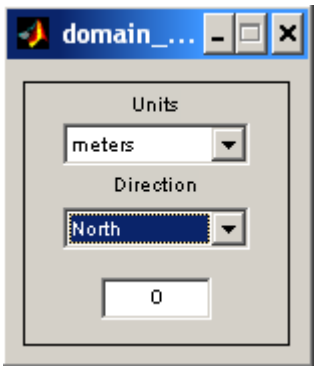
The grid resolution of the QUIC-URB modeling domain is specified in the text boxes on the upper right hand corner of the City Builder window. It is strongly recommended that the grid size should be set before creating buildings. If the buildings are created first, and then grid resolution is changed, the buildings may not be resized optimally. Notice that the grid size can be different in the x, y, and z directions. For example, one may desire more resolution in the vertical and set this to 2 m, while setting the horizontal grid to be 5 m square. The default grid size is 1 meter.

## Domain Size

The domain size (number of grids or distance in the East-West (x), North-South (y) and vertical (z) directions) can be changed using the text boxes on the right hand side of the City Builder window. The units can be toggled between distance (meters) and number of grid cells using the pull-down menu selector.

You can also right click on empty space in the city builder map and add more grid cells to the north, south, east, or west of the domain. This is shown in the figure below.

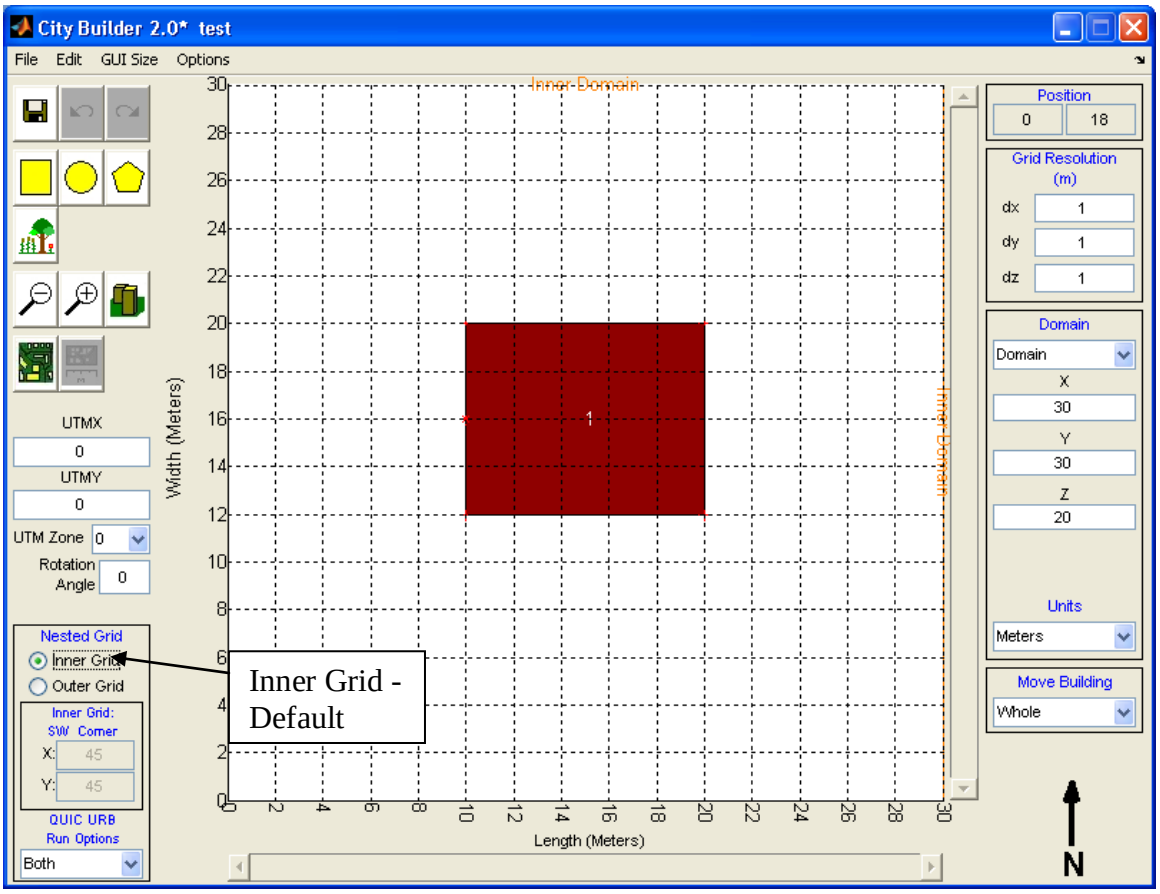


## Inner and Outer Grids

QUIC is normally run with a building-resolved grid. However, to cover larger areas, an outer grid option has been added. Here, the grid cell size is larger and buildings are not resolved. The outer grid could extend for tens of kilometers with 100 meter horizontal grid resolution, for example. Buildings are not put on the outer grid; rather a roughness length parameterization is used instead. While buildings are not used on the outer grid, vegetation canopies (see the following section) may be used on the outer grid to simulate different land use characteristics over large areas. Note that a release of airborne contaminant on the inner grid will automatically be transported onto the outer grid (see QUIC-PLUME section).

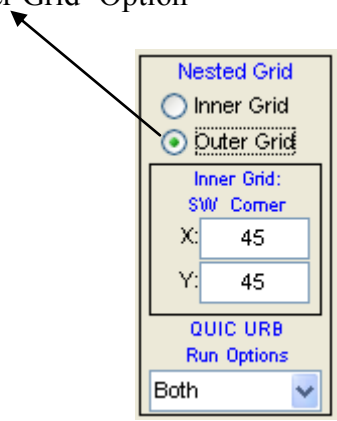
When the City Builder is first brought up, it defaults to the inner grid.



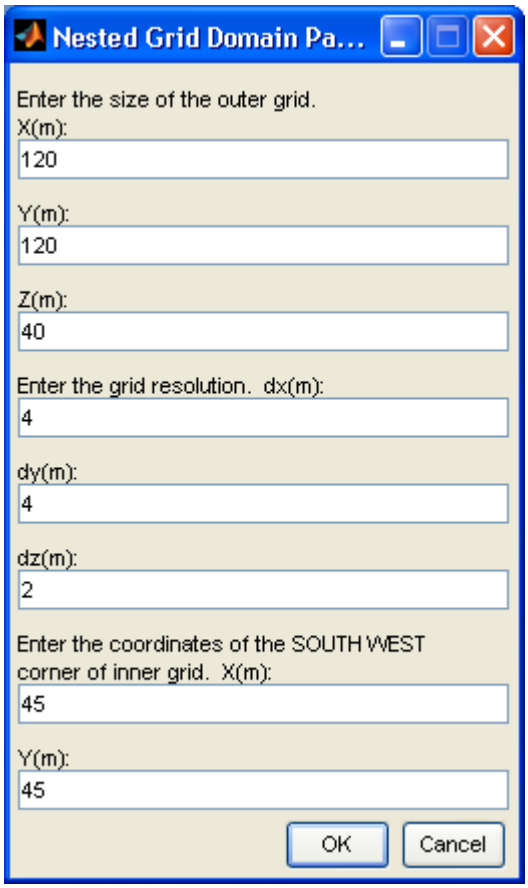
To specify an outer grid, one needs to:

- First save the existing project on the inner grid using the 'save' button

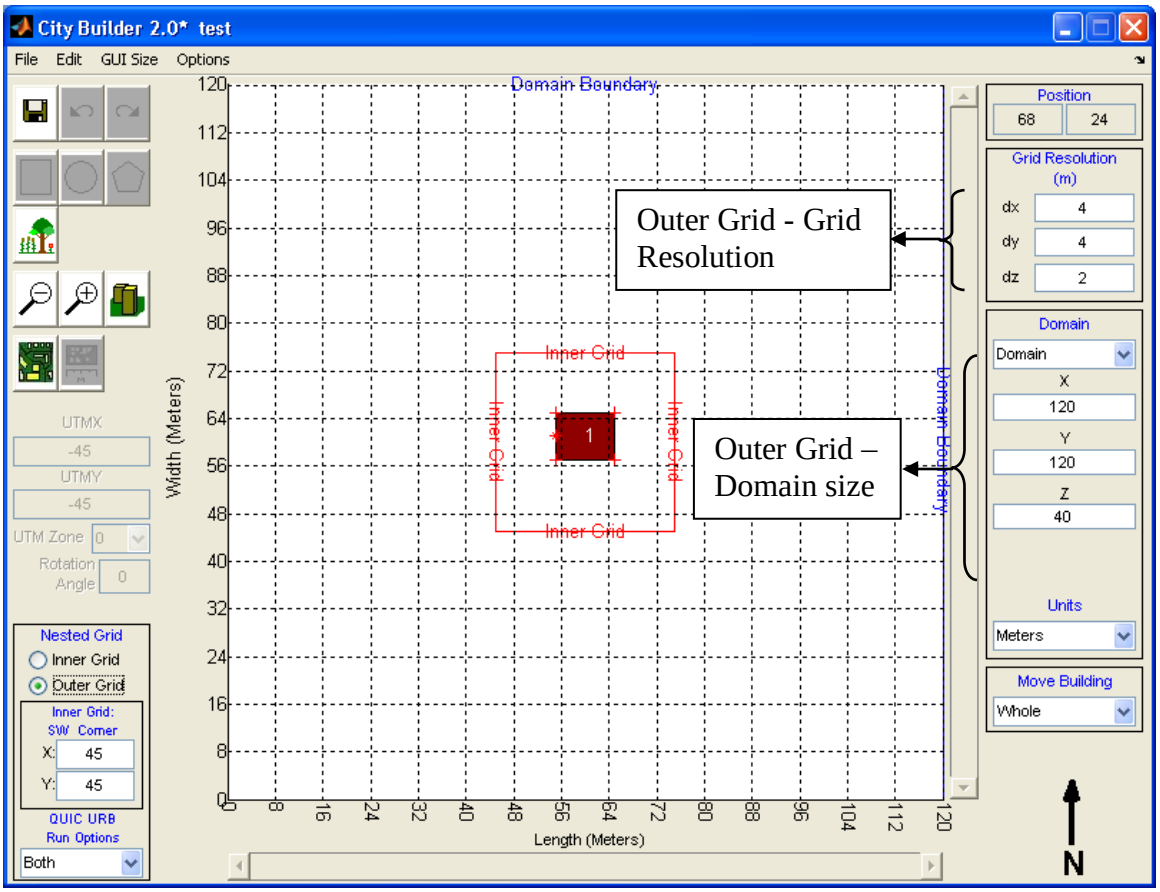
- Now select the 'Outer Grid' Option




- This opens a window where in the outer grid parameters can be entered

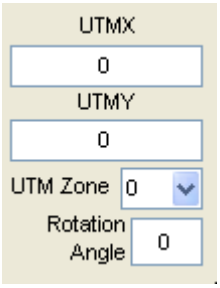



- Type in the appropriate parameters and press 'ok'
- This places an outer grid around the inner grid in the City Builder
- Various parameters of the outer grid such as the grid resolution and domain size can be now be seen in the City Builder




## Geo-Referencing QUIC Domains

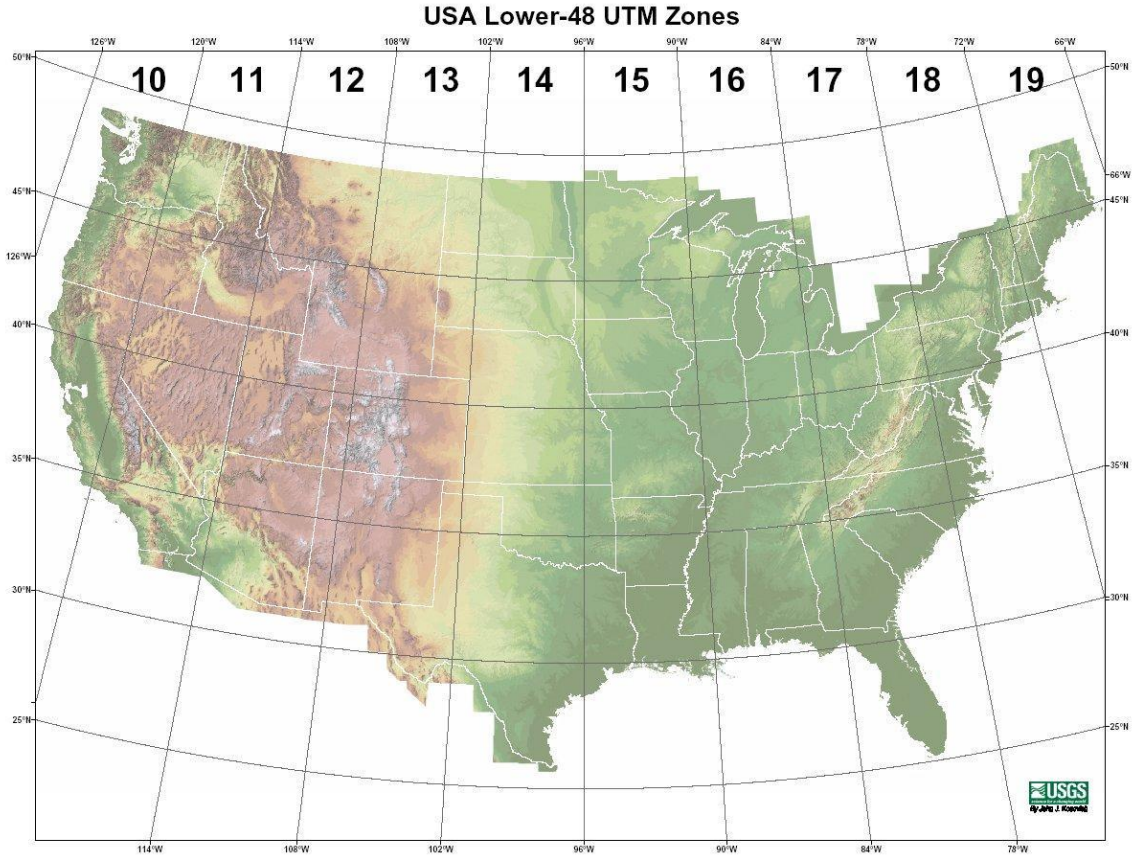
New to QUICv4.5 is the ability to geo-reference domains through the controls on the left-hand side of the GUI shown below:



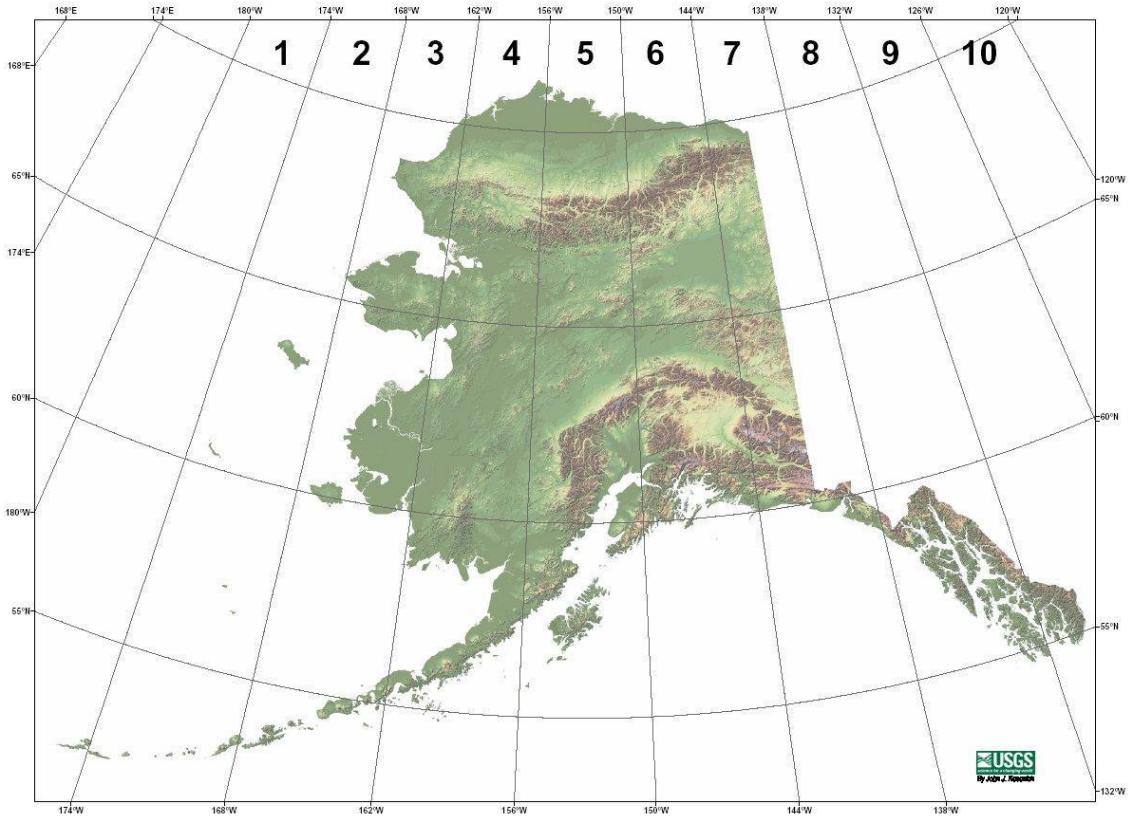

QUIC domains are geo-referenced using the Universal Transverse Mercator (UTM) system. The required inputs are the UTM zone, UTMX and UTMY locations of the inner grid origin, and domain rotation from true north (clock wise rotations of the domain are considered positive angles). The GUI automatically calculates the UTMX and UTMY locations of the outer grid origin given the location of the inner grid, the relative location

of the inner grid to the outer grid, and the rotation angle of the domain. The north arrow in the lower-right-hand corner of the GUI shows the direction of true north relative to the grid. Rotation angles can be between -45 and 45 degrees. The user is encouraged to align the domain such that the majority of the buildings in the domain (or buildings of special interest within the domain) are aligned with the orthogonal grid used by QUIC.

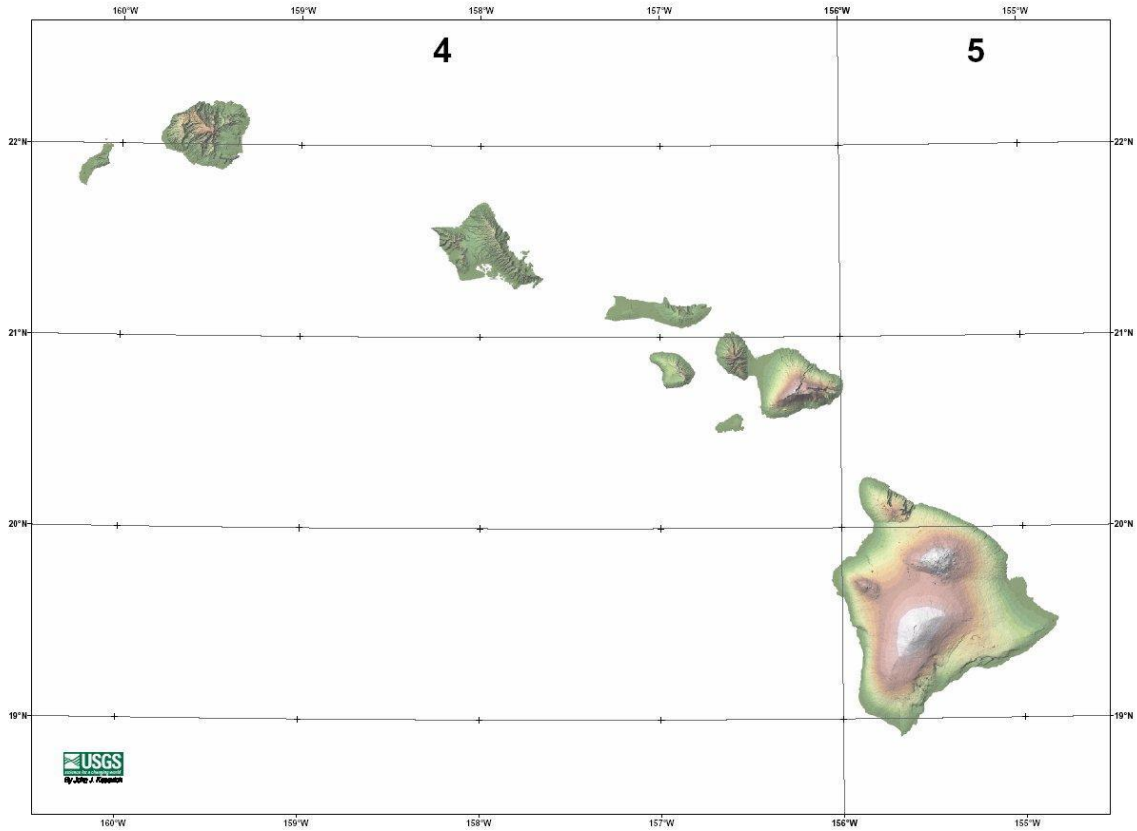
UTM zones range from 1 to 60. The default UTM zone is 0 which is only a place holder since it is not within the range of actual UTM zones. The value of 0 informs the GUI that the QUIC domain has not been geo-referenced. Maps of the UTM zones corresponding the US are shown below:



## Alaska UTM Zones

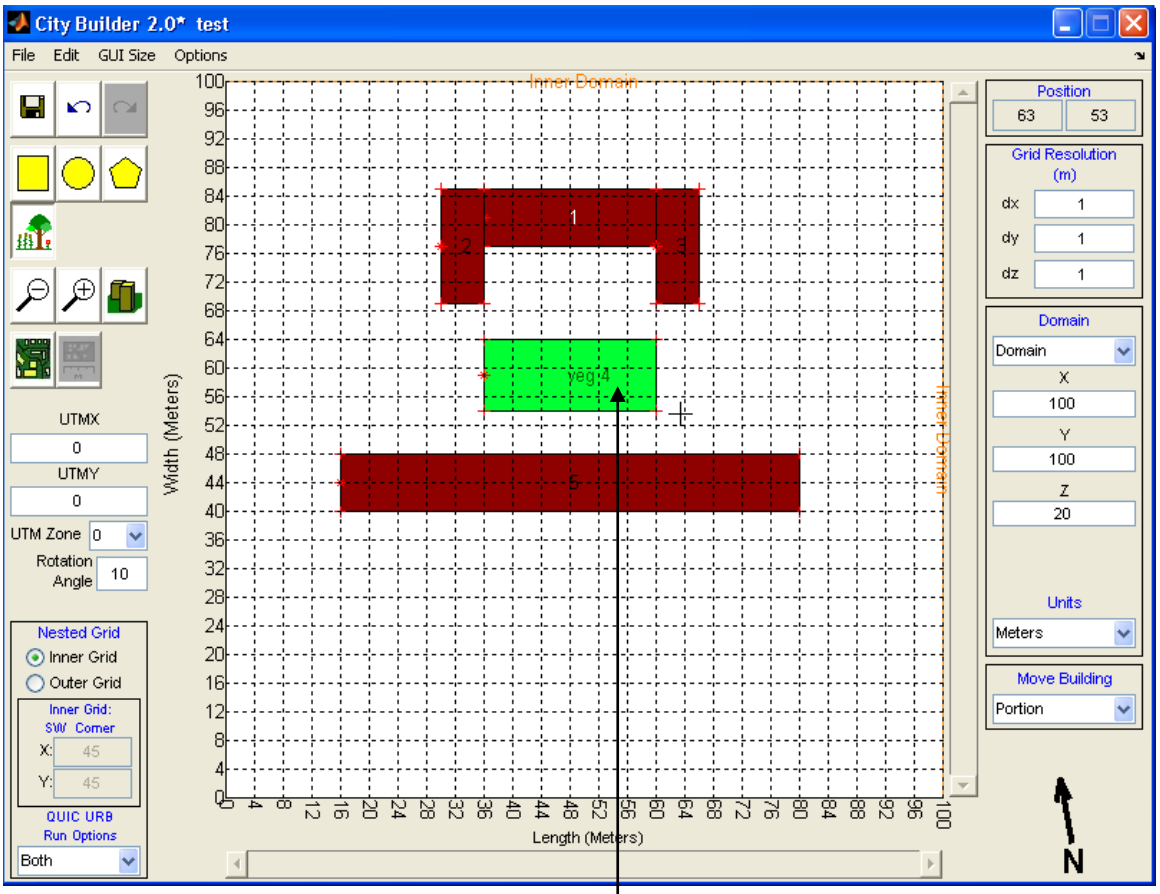



## Hawaii UTM Zones



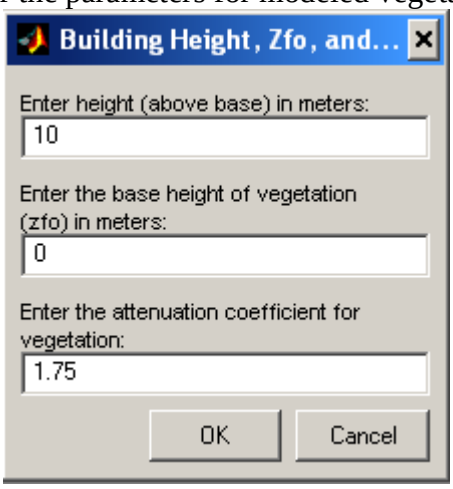
## Vegetation

City Builder provides users the option of placing vegetation in and around buildings. To exercise this option, press the vegetation button . Vegetation canopies are defined in the exact same manner as rectangular buildings with the exception of having an additional parameter that must be defined, the attenuation coefficient following Cionco (1966) as is shown below. The vegetation algorithm acts to attenuate the initial velocity within the defined volume before any of the other building algorithms are applied. These attenuated velocities may be overwritten by some building algorithms that are added later.




Area where the initial velocities will be affected by the presence of vegetation.

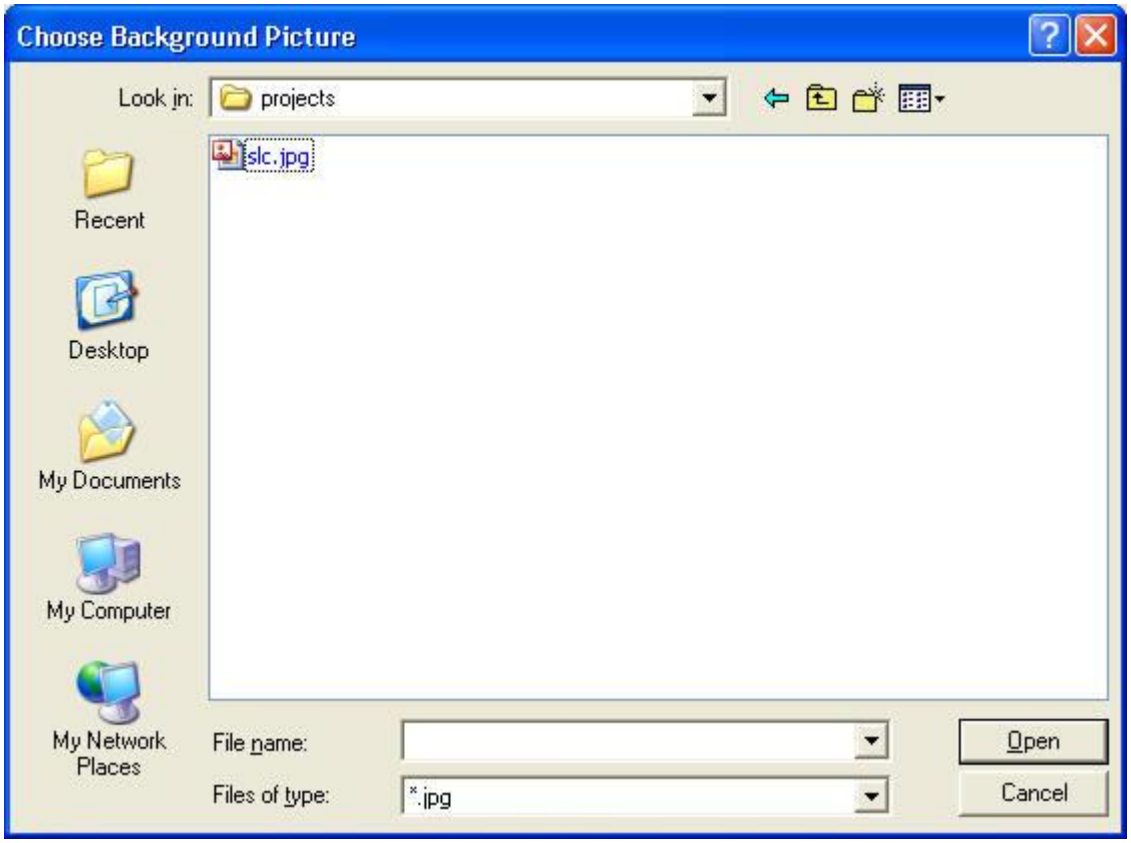
In the pop up window, enter the parameters for modeled vegetation as shown below



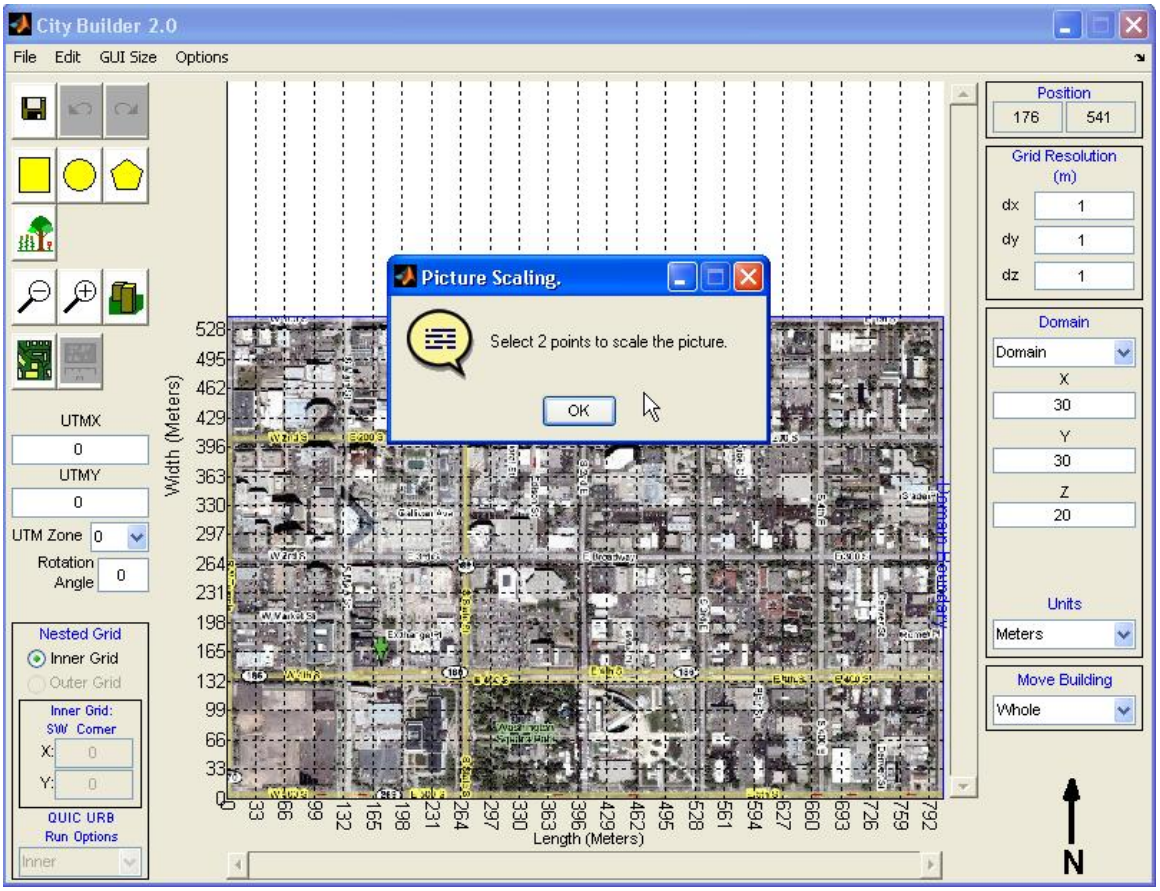

## Add a Background Map

Background maps facilitate more accurate reproduction of urban areas.

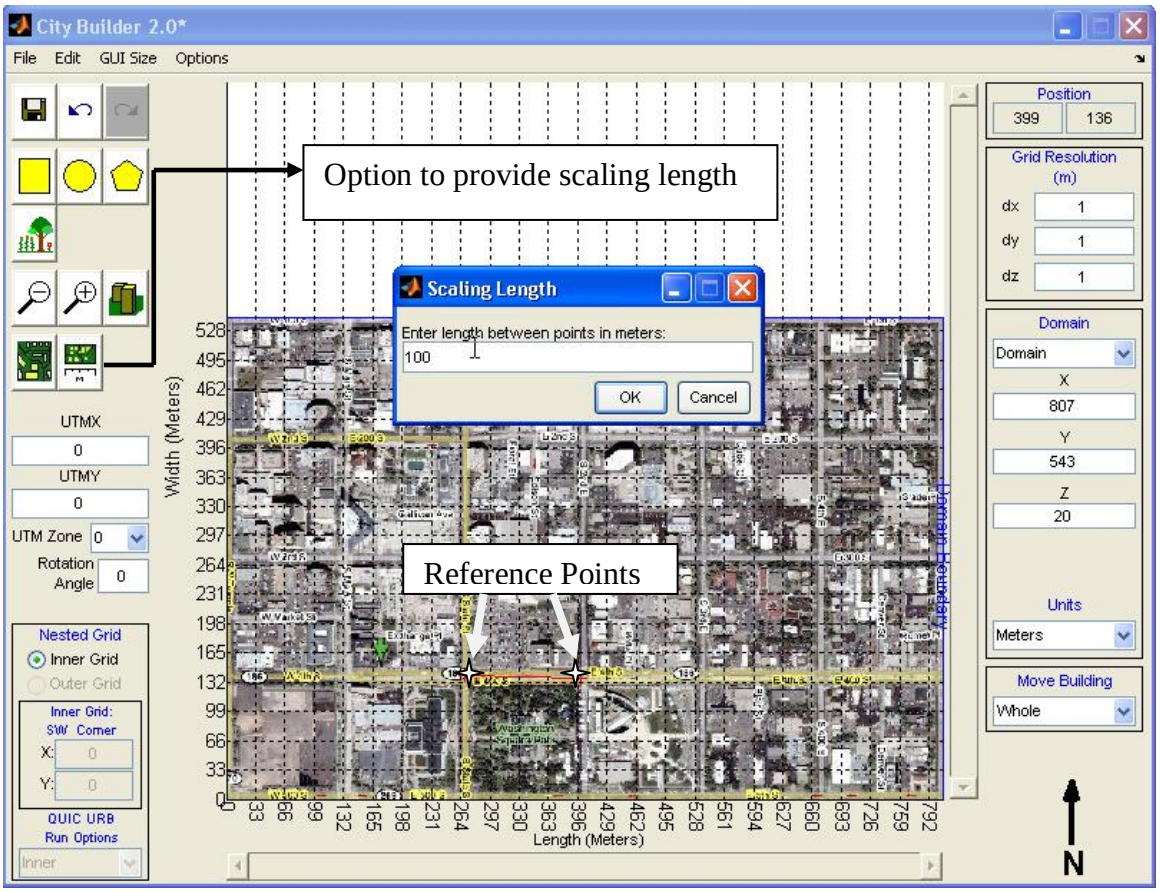
Pressing brings up the prompt to load a JPEG background map seen below.



After a background map has been selected it will appear in the grid and the user will be prompted to select two points with a known distance between them to set the horizontal scaling.

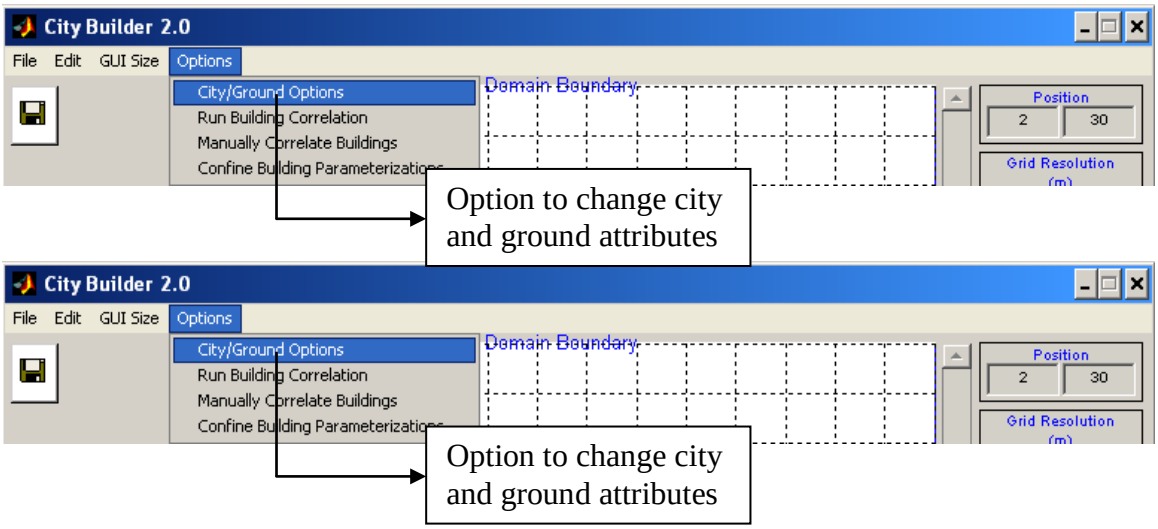


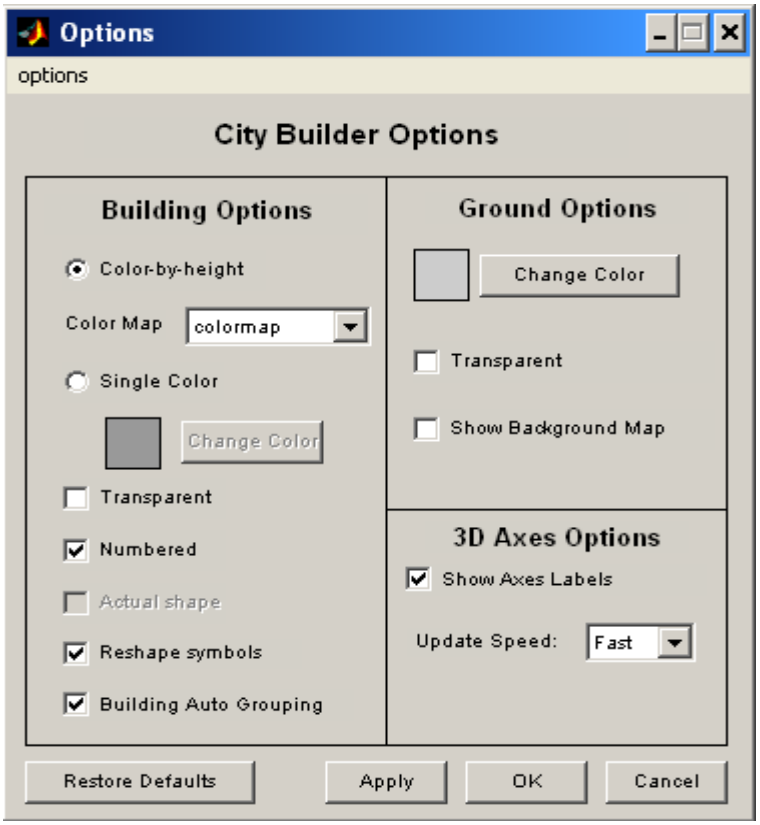
Left-click on two points on the background map that are a known distance apart in meters. Two street intersections (shown here in white) were specified as being 90 m apart. This will automatically scale the grid to the image.



## City Builder Options

Select <Options> on the City Builder pull-down menu to change City Builder viewing options.



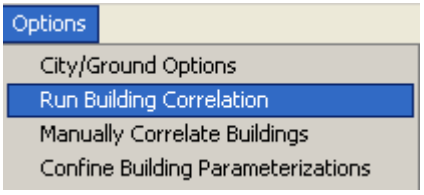
The City Builder Options window controls the color and the information displayed. Building color, ground color, and transparency is specified here. Building numbers and axes can be turned on and off.

- click Apply or OK to accept changes.

Note: The output of VIS Gui is also controlled by this pop-up window.

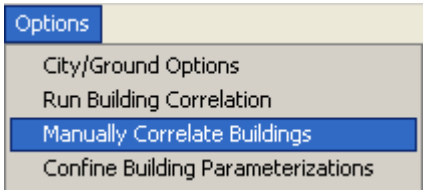
**Run Building Correlation**

Select the 'Run Building Correlation' from the options pull down menu. The run building correlation option looks for buildings that are touching and groups them together through a building group id.



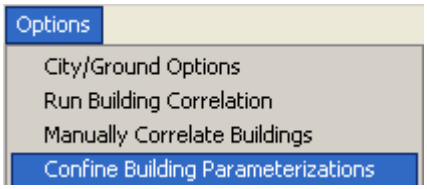

**Manually Correlate Buildings**

Select the 'Manually Correlate Buildings' from the options pull down menu. This option provides the user the capability of manually grouping buildings together, even those that are not touching. This option is not functional in the present version and will be included in the future versions of QUIC

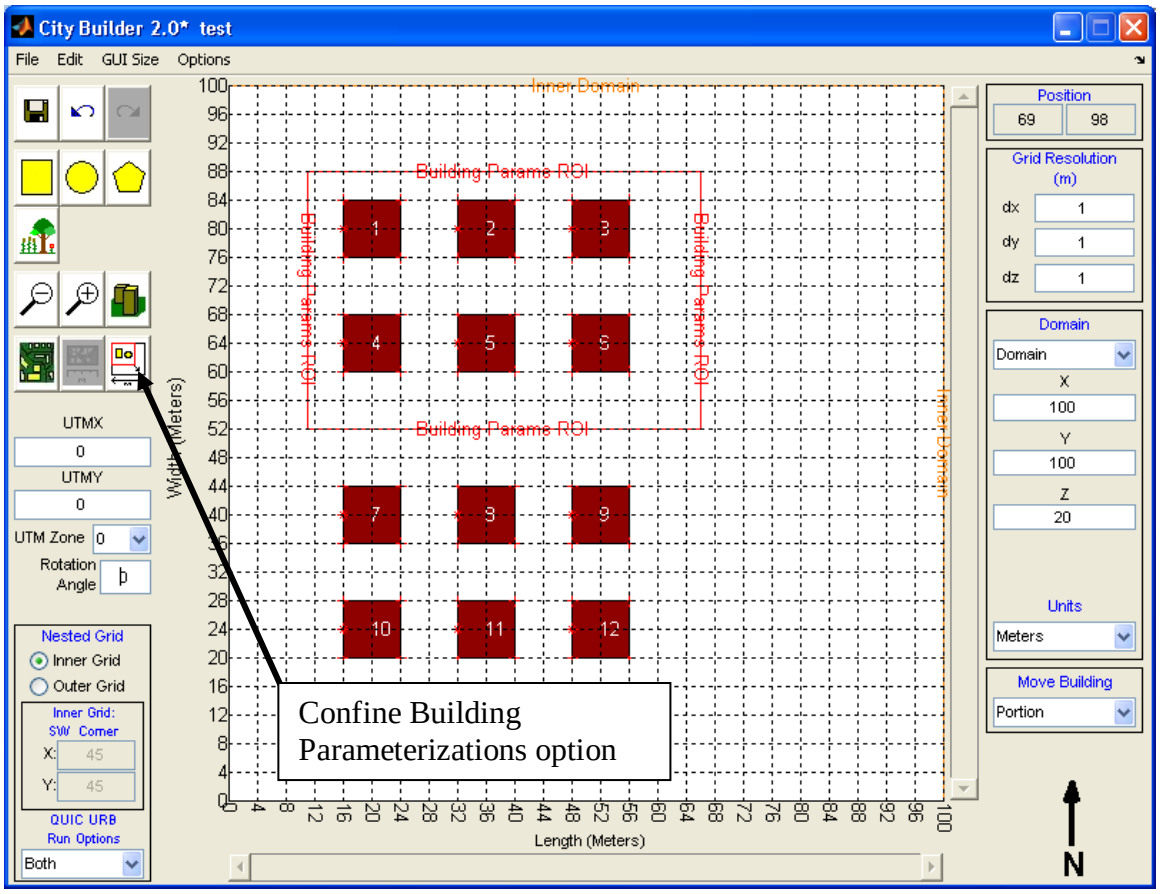



**Confine Building Parameterizations**

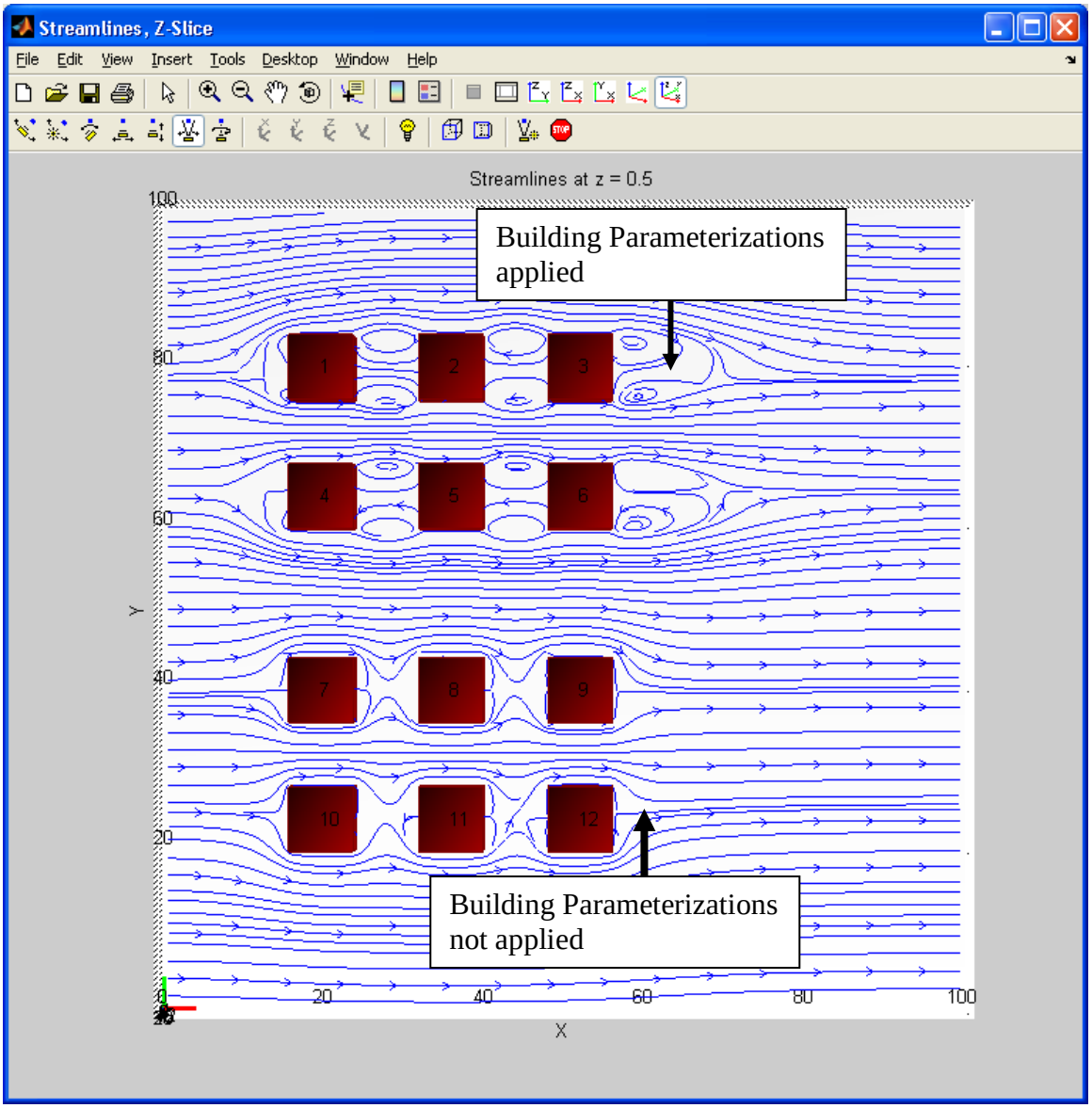
Select the 'Confine Building Parameterizations' from the options pull down menu. This option provides the user the option of applying the building flow parameterizations to only selected buildings, i.e., those within a user defined rectangle. On selecting the option, an icon appears in the main City Builder window as shown below.



Press this button and with the cursor select an area around the buildings where you would like to confine the parameterizations. This is has been shown below:

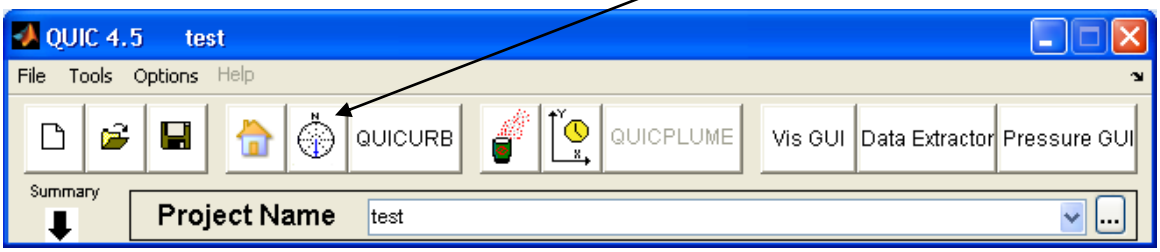


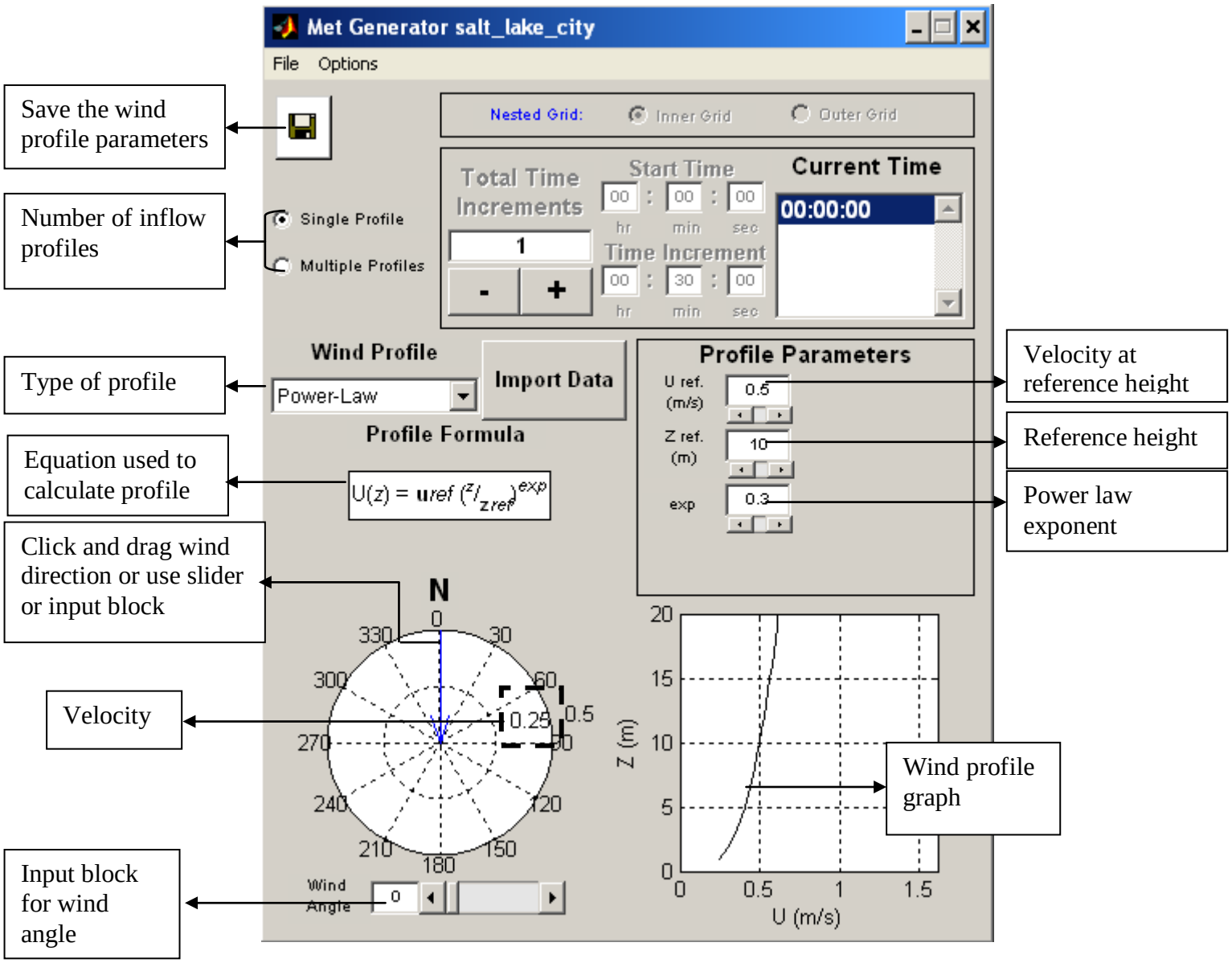

The figure above shows the region over which the building parameterizations are confined. Building flow parameterizations are not applied to the other buildings and only the initial flow field is utilized. The results after applying mass conservation are shown in the figure below:


Streamlines, Z-Slice
File Edit View Insert Tools Desktop Window Help
Streamlines at z = 0.5
Building Parameterizations applied
Building Parameterizations not applied
1
2
3
4
5
6
7
8
9
10
11
12
100
80
60
40
20
0
20
40
60
80
100
X
Y

# Met Generator

To open the Met Generator click on the wind rose icon

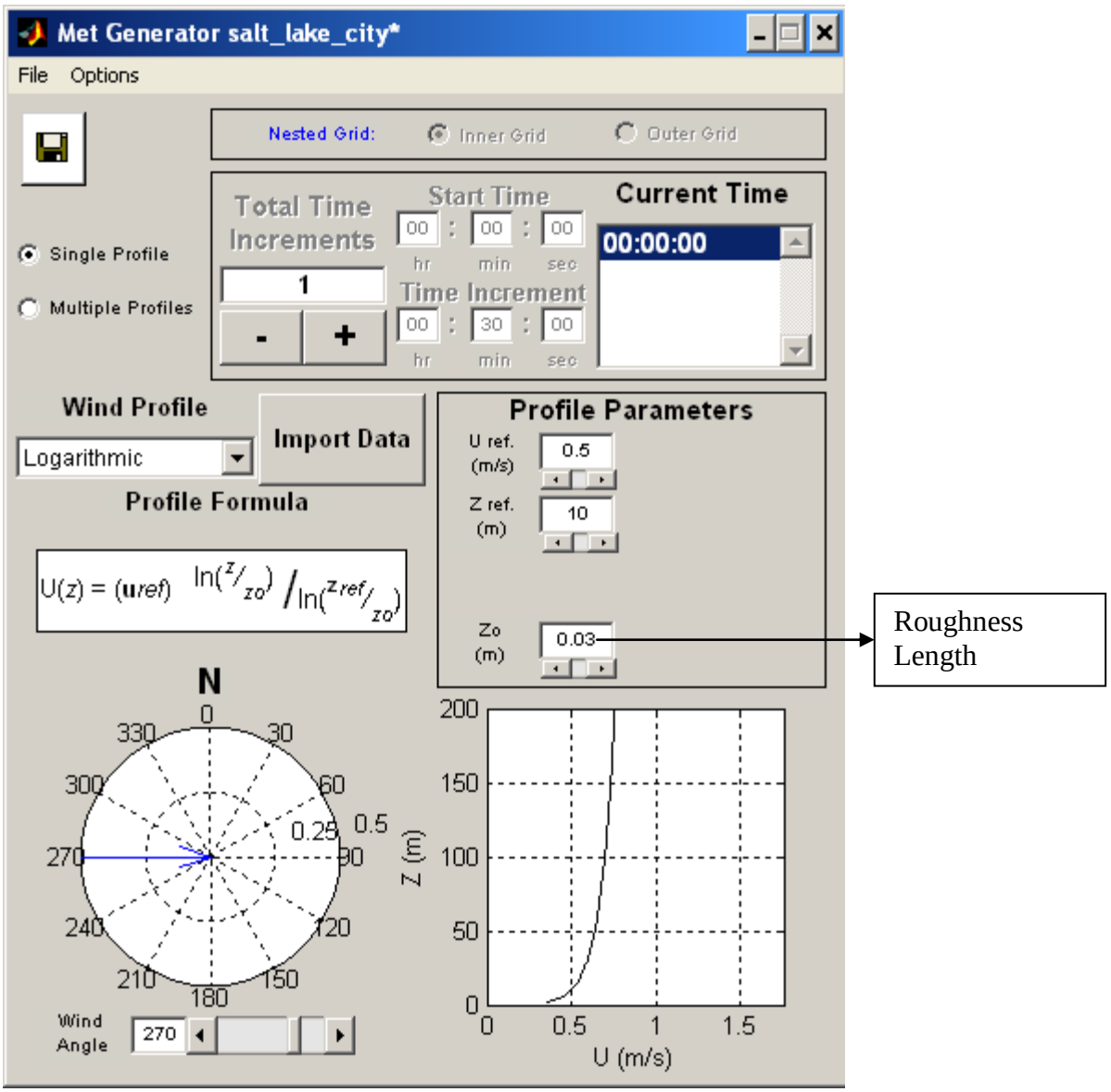


- The inflow conditions are specified here. The wind direction is specified through the compass and arrow. The wind speed is a function of height and can be specified as a power-law, log-law, urban canopy, or user-specified profile.

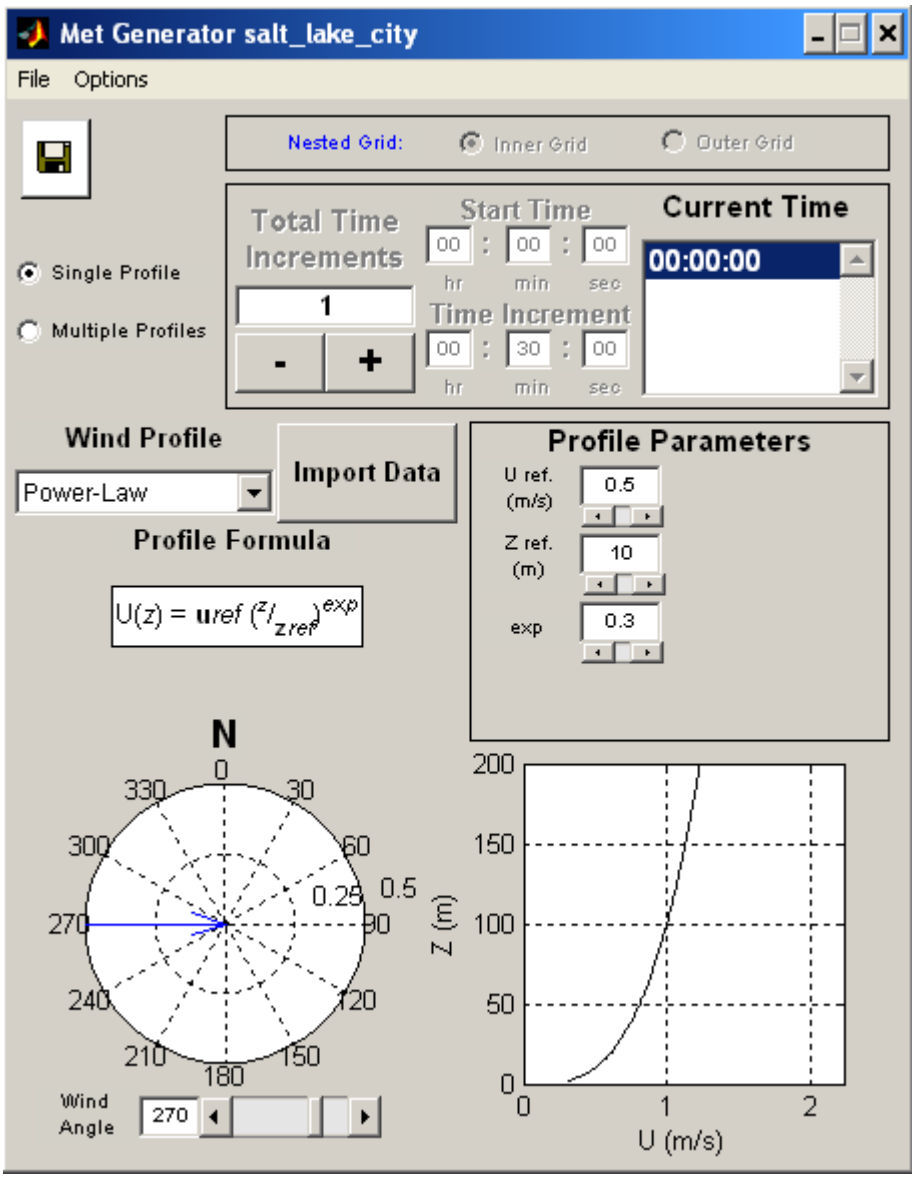
## Logarithmic Profile

The logarithmic wind profile equation is shown when Logarithmic is selected.



## Power-Law Profile

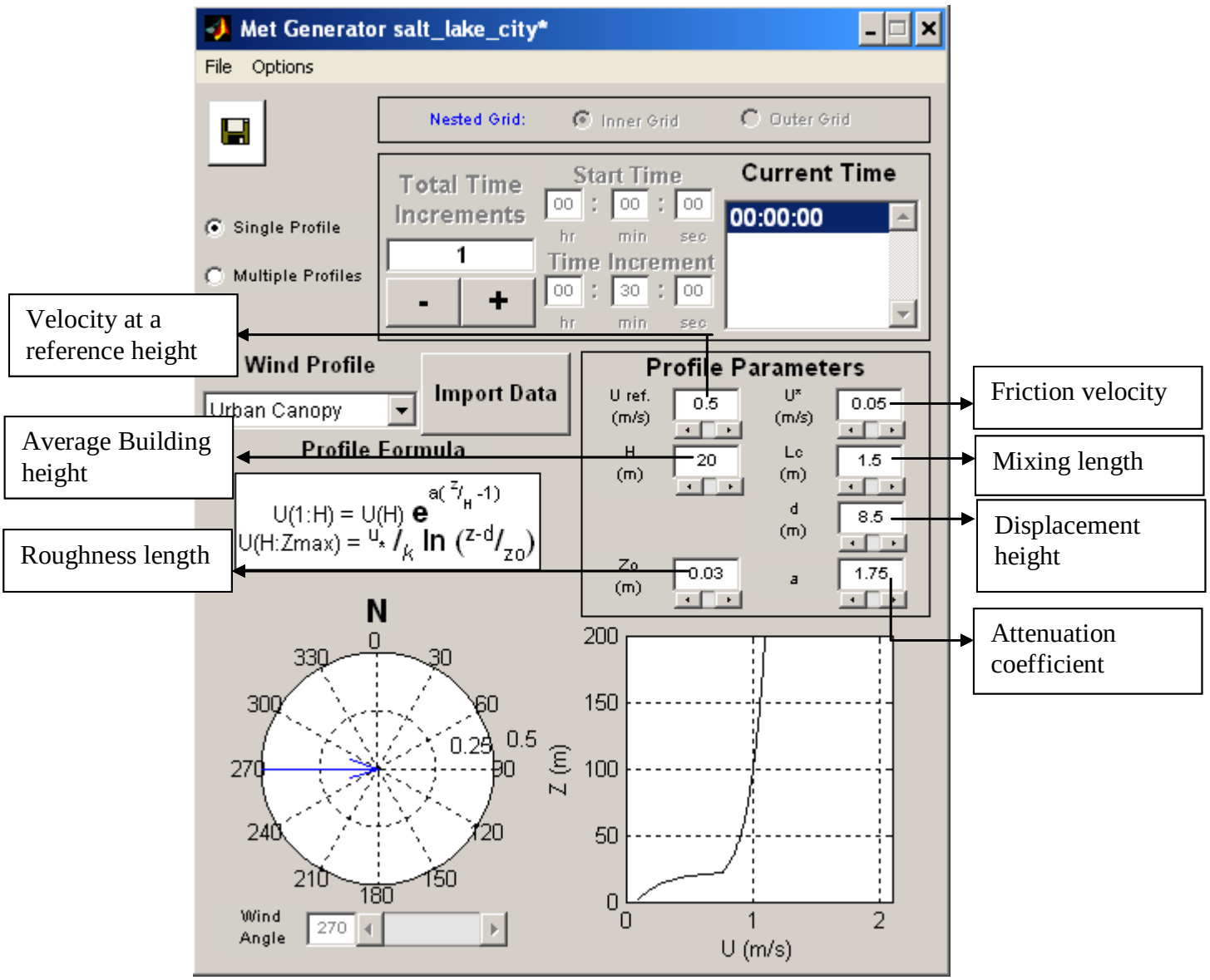
The screenshot below shows the power-law profile parameters.




The wind profile type is selected through the "Wind Profile" pull-down menu. The profile parameters box to the right changes for each wind profile type.

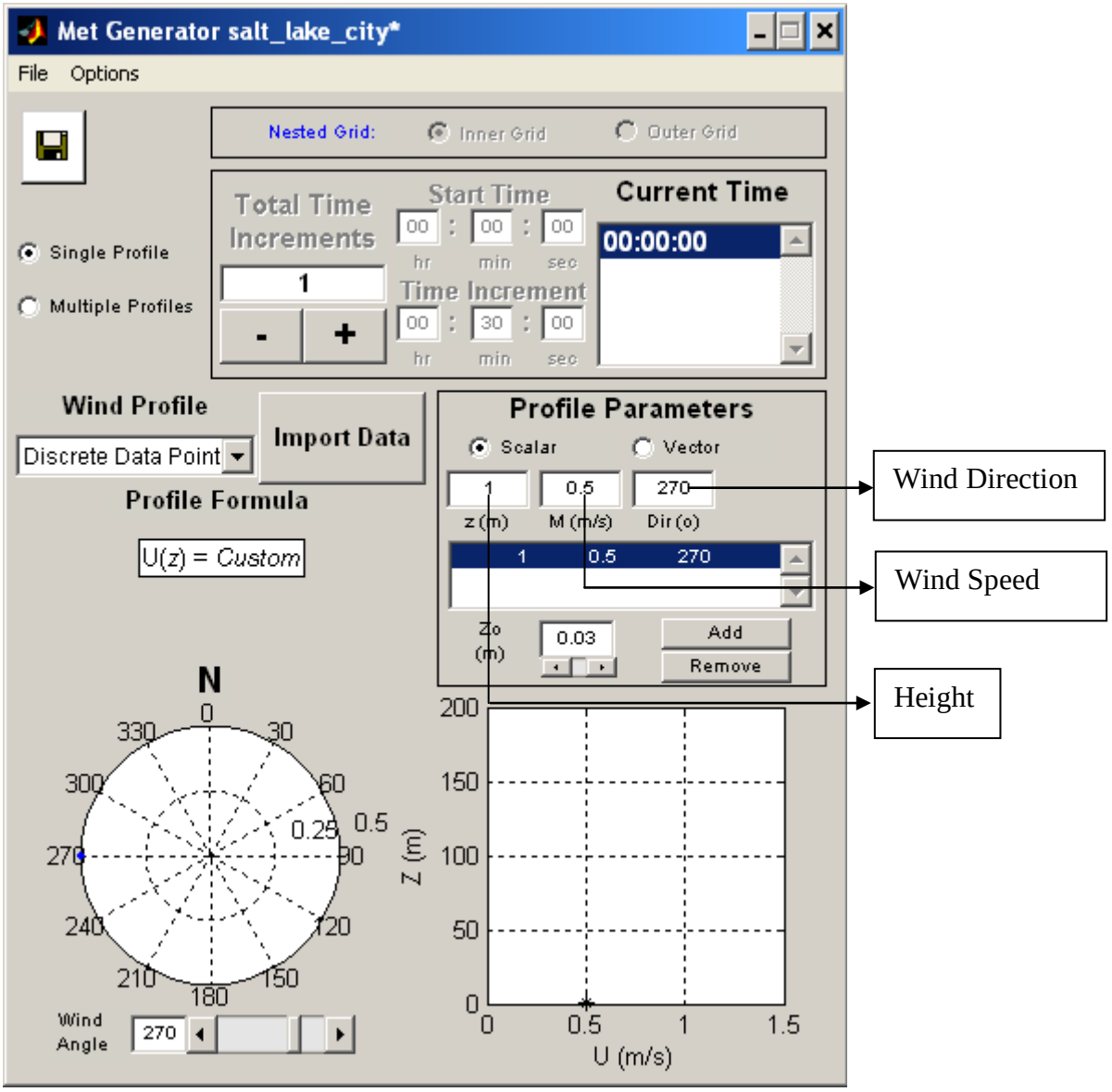
## Urban Canopy Profile

In the screen shot below the Urban Canopy Profile has been selected. This profile represents the area-average impact of buildings (or vegetation) on the flow. The region of low wind speeds below average canopy height H is typical of winds that have been slowed by buildings and vegetation upstream of the point of measurement. This profile uses a different equation for winds above and below the reference height (H). As seen in the screen shot this profile requires a lot more parameters than any of the other profiles. It is possible to create a profile that is discontinuous at H, thus care should be taken to avoid this situation, as it will affect the accuracy of the results.



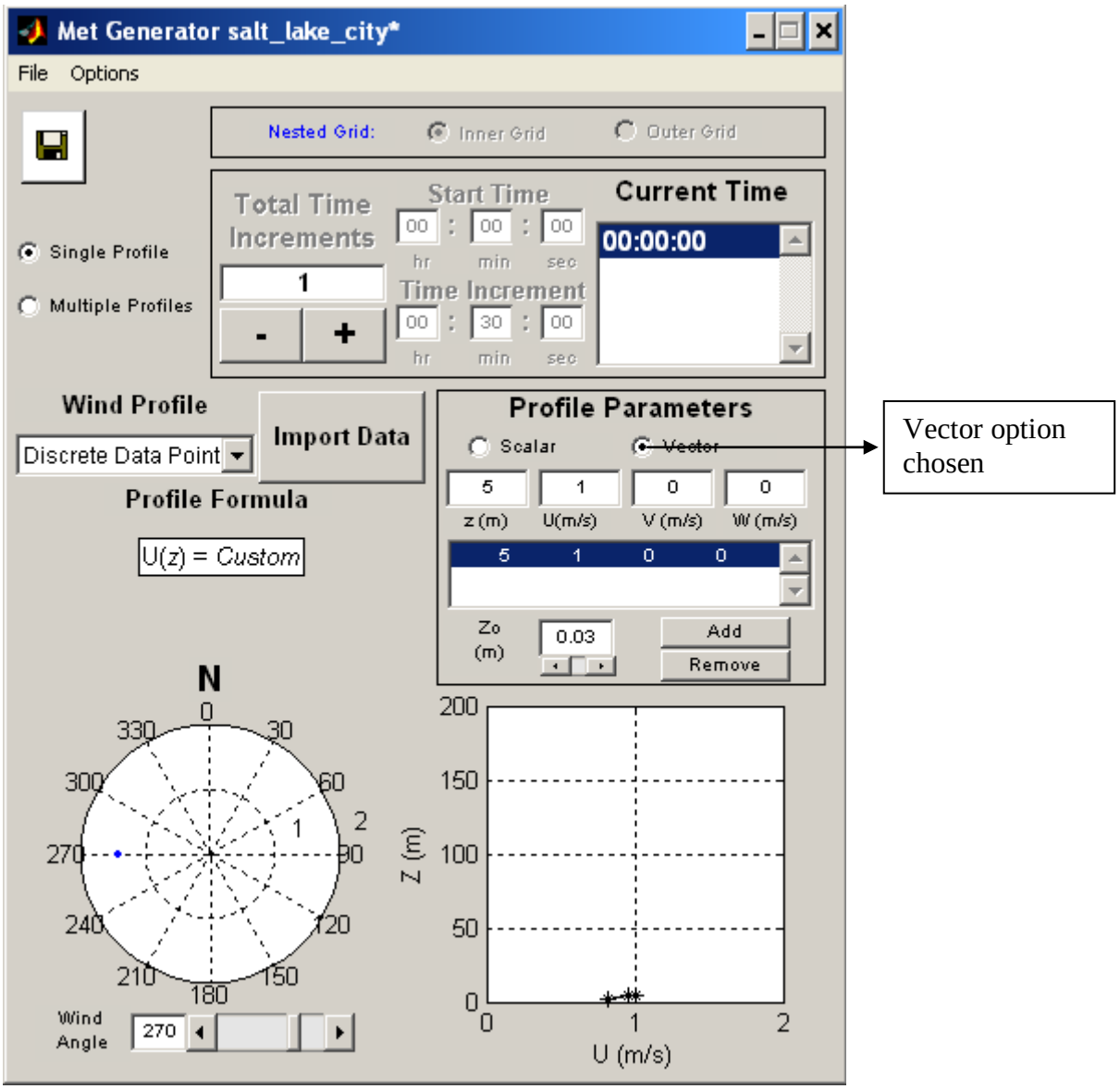
## Discrete Data Points (User-Specified) Profile

A text box will come up where you can hand type in information. Using this profile it is possible to enter profiles with directional shear (i.e., wind direction does not have to be constant with height). Users are cautioned against running the model with large amounts of directional shear below building height as this will likely produce unrealistic results.




In the text box shown in the figure above, enter the value of wind speed at a given height. The wind direction and roughness length z0 can also be specified.

If the user has the values of all three components of velocity at a given height, the user can input them manually by choosing the vector option as shown below. It should be noted, however, that QUIC-URB does not currently use the vertical velocity components when initializing the wind field. The profile is extrapolated from the lowest data point to the ground using a logarithmic profile with a user specified roughness length ($z_0$).

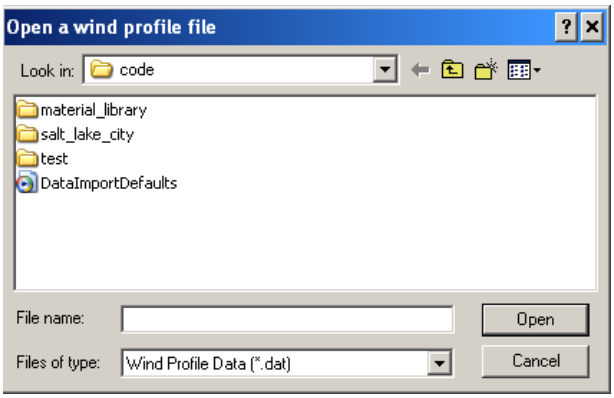


## Import Data

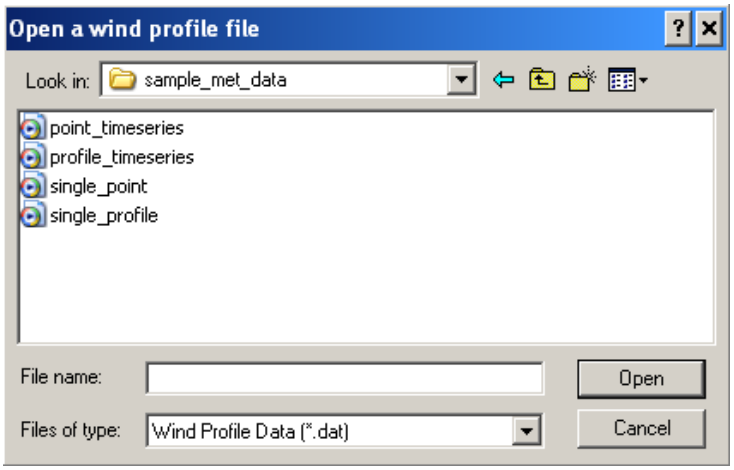
One can also directly import a data file in a four column space delimited format: time (decimal hours), height (meters), wind speed (meters per second), wind direction (degrees). To do so

- select "Import Data"


This opens a window titled 'Open a Wind Profile' as shown below

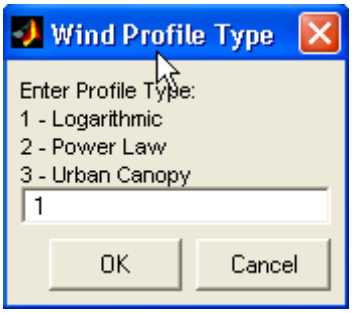



Open the folder titled 'sample_met_data' in the main QUIC folder. This folder has sample velocity profiles.

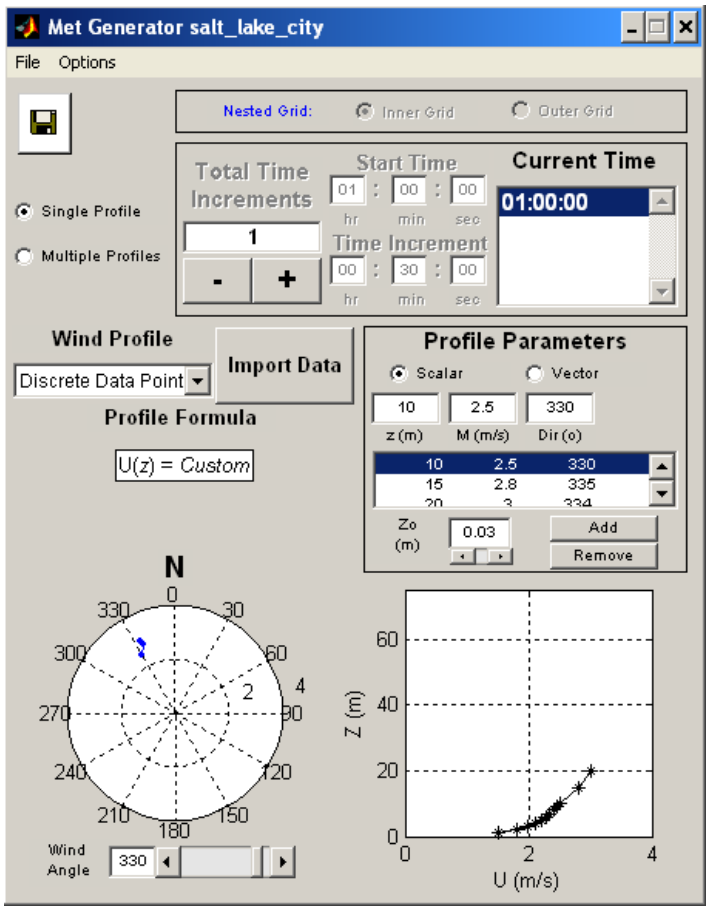



- select a wind profile data file in the format mentioned in the above paragraph
- press "Open"
- The GUI will automatically determine if the imported data file constitutes a profile or single point measurement.

- If the data is for a single point measurement, the GUI will prompt the user to select a profile type as seen below. Enter the number corresponding to the desired profile type and press “OK”.

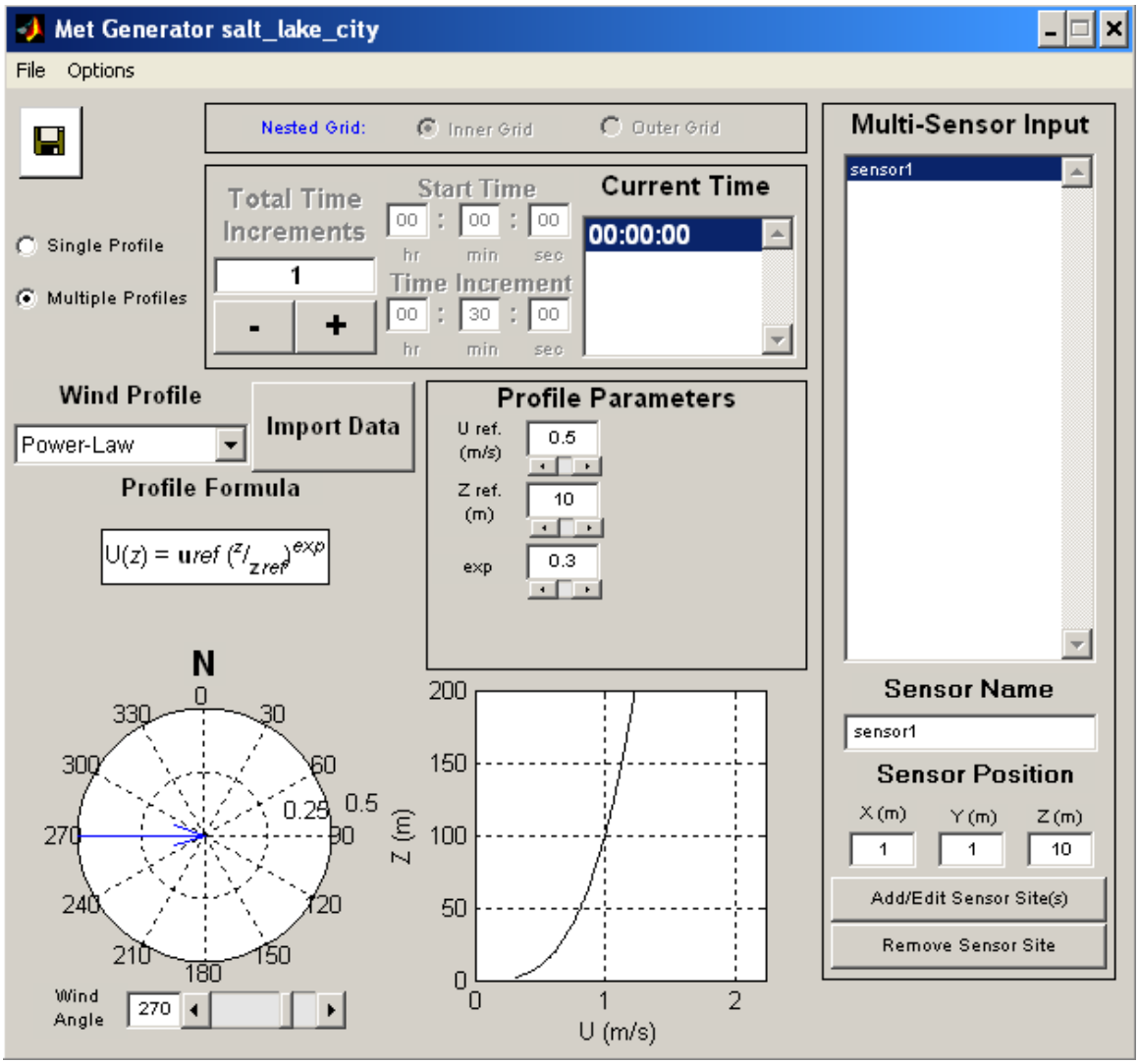


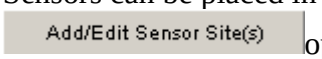

- Figure below shows a sample wind profile imported into the Met Generator



## Multiple Profiles

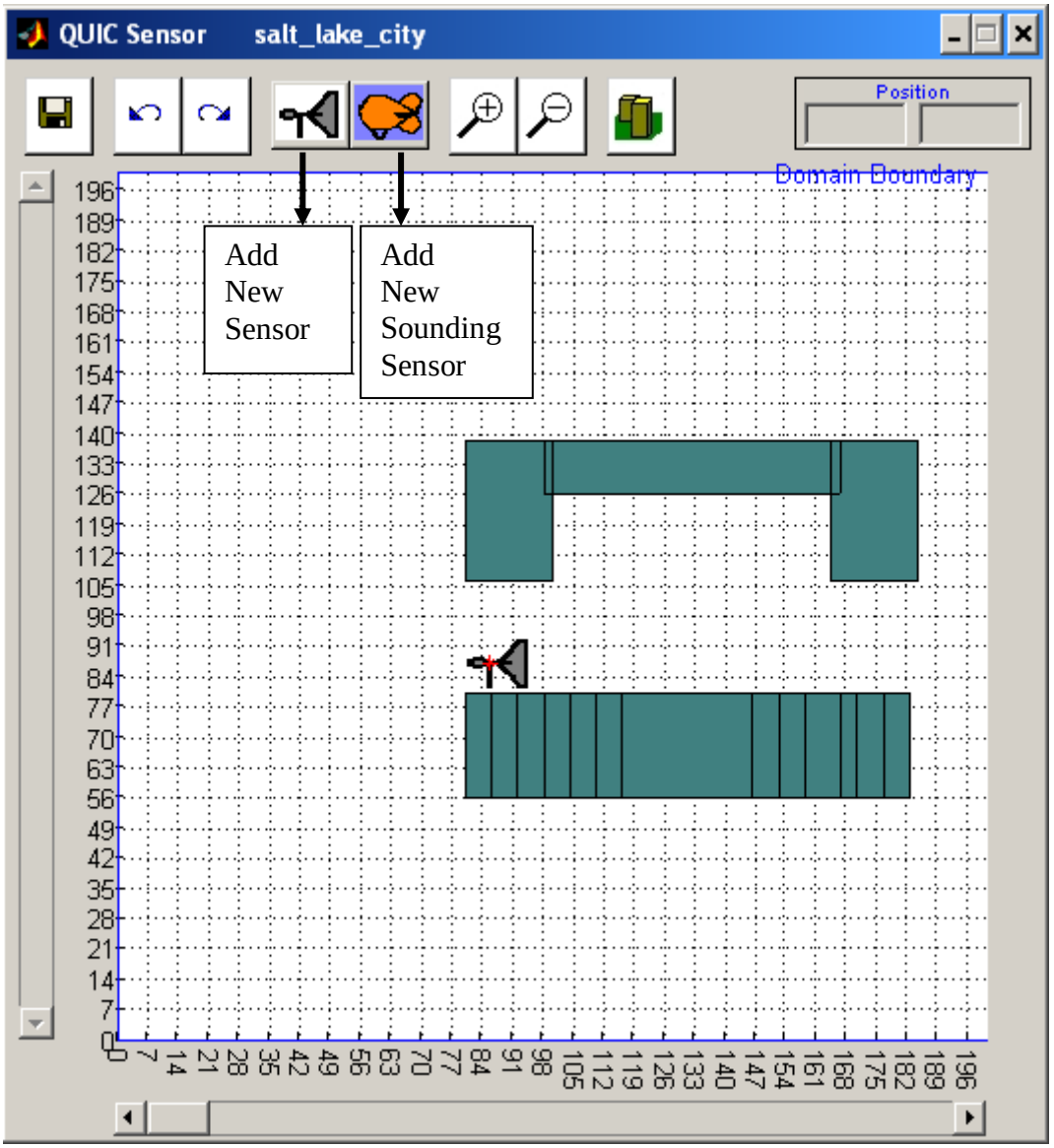
To add multiple profiles, choose the 'Multiple Profiles' option as shown.




QUIC-URB uses quasi-3D Barnes mapping to interpolate the initial wind fields between sensors. This option is useful for modeling real cities that have multiple wind sensors in different parts of the city. Sensors can be placed in the modeled city by choosing the 'Add/Edit Sensor Site(s)' 

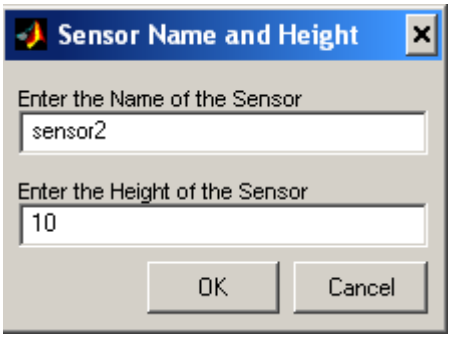
option.

Clicking this option brings up the 'QUIC Sensor' Window as shown.

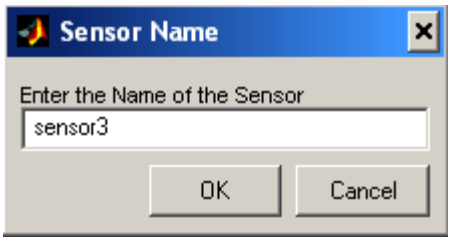



The QUIC Sensor window has the 'Add New Sensor' (single point measurement with an extrapolated profile) and 'Add New Sounding Sensor' (profile measurement with logarithmic extrapolation to the ground) options as shown. To place a sensor at a particular location, click in the appropriate icon and place the sensor with the cursor by clicking at the desired location as shown.

This brings up a pop up window as shown. In this pop up wind enter the sensor name and the height of the sensor and press 'ok'. This places the sensor at the desired location. This is shown below.

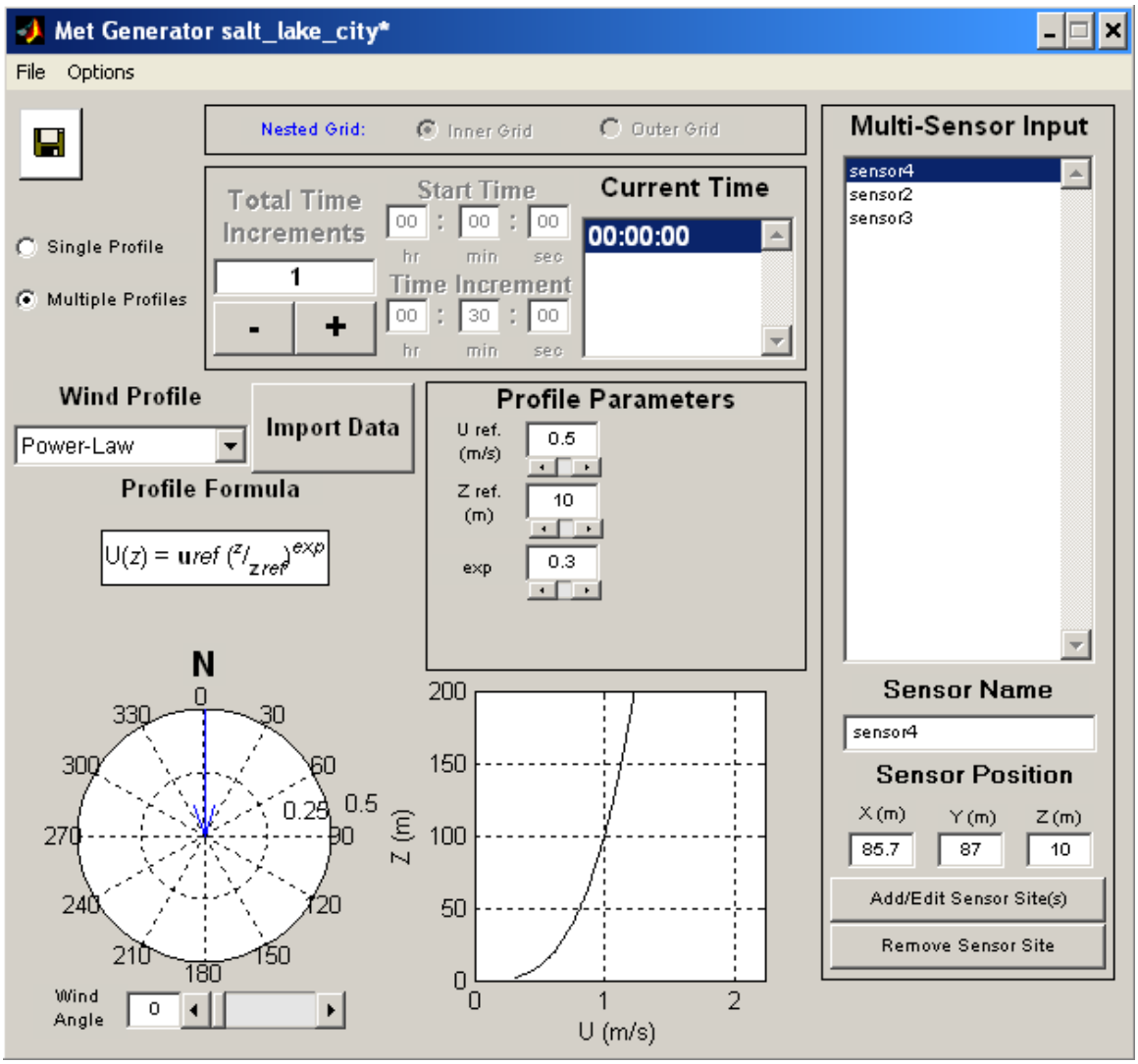


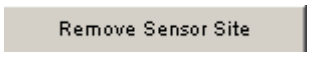

To place a sounding sensor, follow the same procedure and a pop up window will appear. In this pop up window enter the sensor name and press 'ok'. This will place the sounding sensor at the desired location.




After all the sensors are placed, click the save button before exiting.

Once the user exits the QUIC Sensor window, he / she is directed back to the MET Generator. Here, select the appropriate sensor and enter the wind parameters.

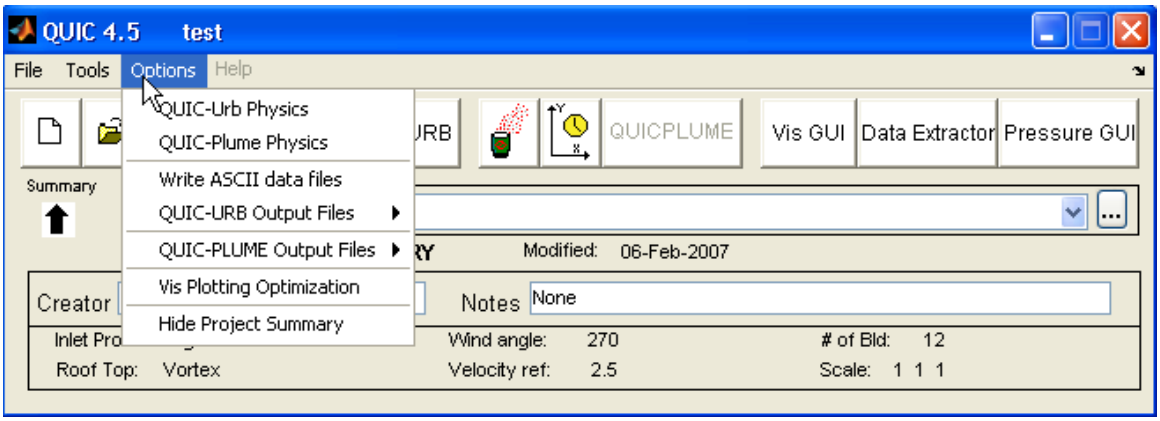



To delete a sensor, use the 'Remove Sensor Site' 

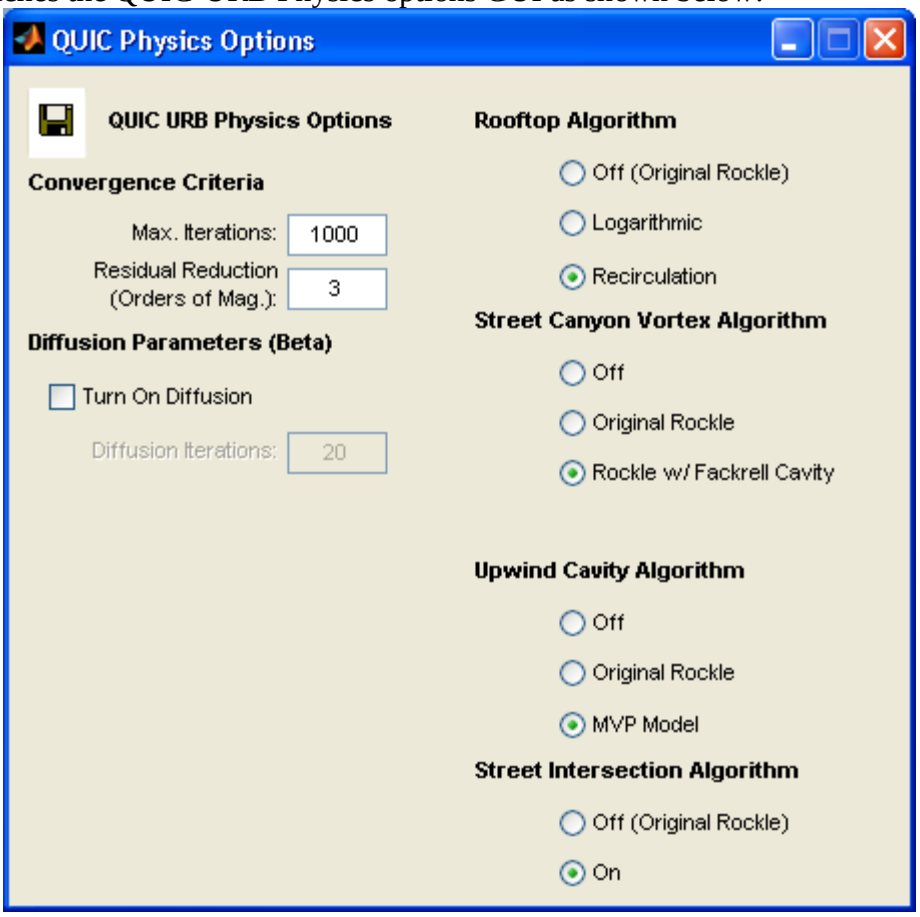
option.

# QUIC-URB Physics Options

Several empirical algorithms used by the QUIC-URB wind model can be changed through the Physics Options pop-up window. On the main QUIC-GUI window select <Options> <QUIC-URB Physics>.




This launches the QUIC-URB Physics options GUI as shown below:



## Rooftop Algorithm

There are three choices for the rooftop algorithm: original Rockle (a free-slip boundary condition), logarithmic (a no-slip boundary condition), and recirculation (a rooftop vortex). The recirculation option is recommended.

## Street Canyon Vortex Algorithm

There are two choices for the street canyon vortex algorithm: original Rockle and Rockle with Fackrell Cavity. The Rockle with Fackrell cavity is recommended.

## Upwind Cavity Algorithm

There are two choices for the upwind cavity algorithm: original Rockle and MVP Model. The MVP model is recommended.

## Street Intersection Algorithm

There are two choices for the street intersection algorithm: off (original Rockle) and on. Street intersection turned on is recommended.

## Convergence Criteria

The maximum number of iterations to perform to solve the Poisson equation used for mass conservation. This forces the code to stop even if the residual convergence criterion has not been reached. The recommended maximum number of iterations is between 500 and 1000.

The order of magnitude reduction in the residual to reach convergence. Three orders of magnitude reduction is recommended. Smaller problems generally require fewer iterations to converge.

## Diffusion (Beta)

This option imposes the diffusion operator on the velocity field. This attempts to mimic the effects of turbulent diffusion on the mean flow field. The number of iterations determines how many times it is implemented. This option is experimental and as such is turned off by default.

- After making selections, save the current profile by clicking the save button.

# Running QUIC-URB

Once the building layout and inflow wind conditions have been defined, the QUIC-URB code can be run (if the parameters are not specified, the QUIC-URB button will be deactivated).

- Select the QUIC-URB

icon. The light bulb indicates that QUIC-URB is running and the wind fields are being computed.

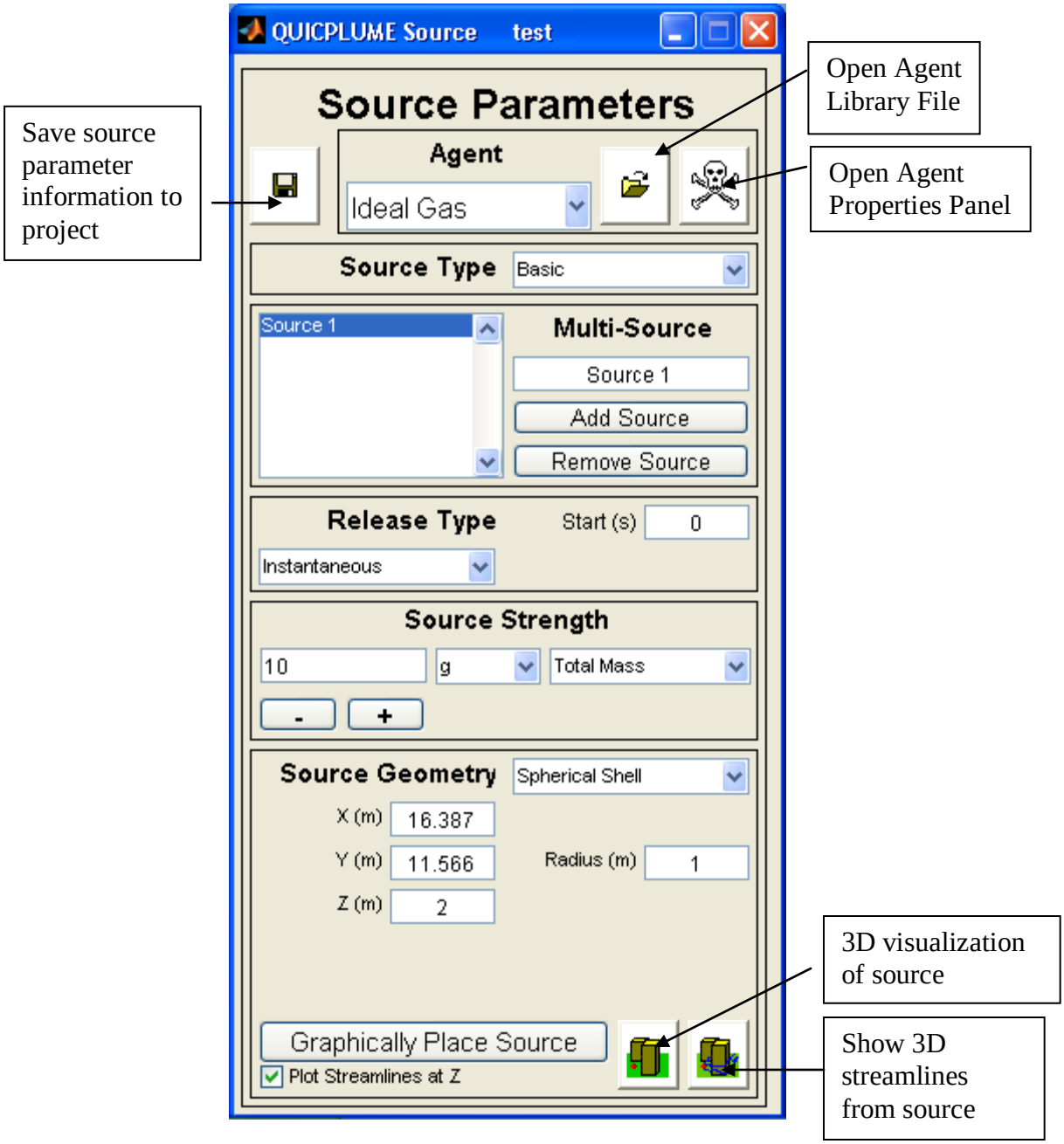
When the light bulb disappears, the QUIC-URB model has finished running. The computed wind fields can be used as input to the QUIC-Plume dispersion model, the QUIC-Pressure code, or can be visualized with the Vis-GUI. Refer to the Vis-GUI section for visualization options.

# QUIC-PLUME

# QUIC-PLUME Source Parameters

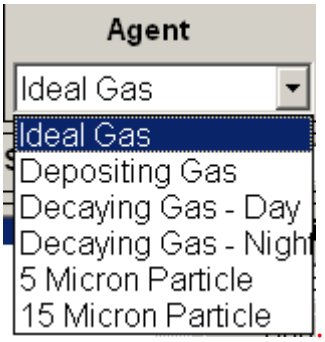
- To open the QUIC-PLUME Source Setup window click on the smoke stack icon.

- The source type, the release location, the size of the release, the release duration, and the material and associated properties are set here.



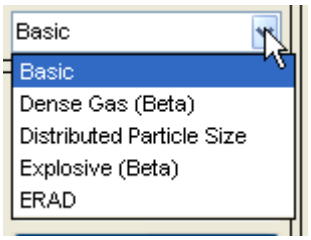
## Agent Type

The artificial agent library is included in the standard release of QUIC. This library includes six different agent types with various properties which can be used in QUIC-PLUME. They are shown below. An Official Use Only agent library with actual agent properties is available for users who meet the release requirements.



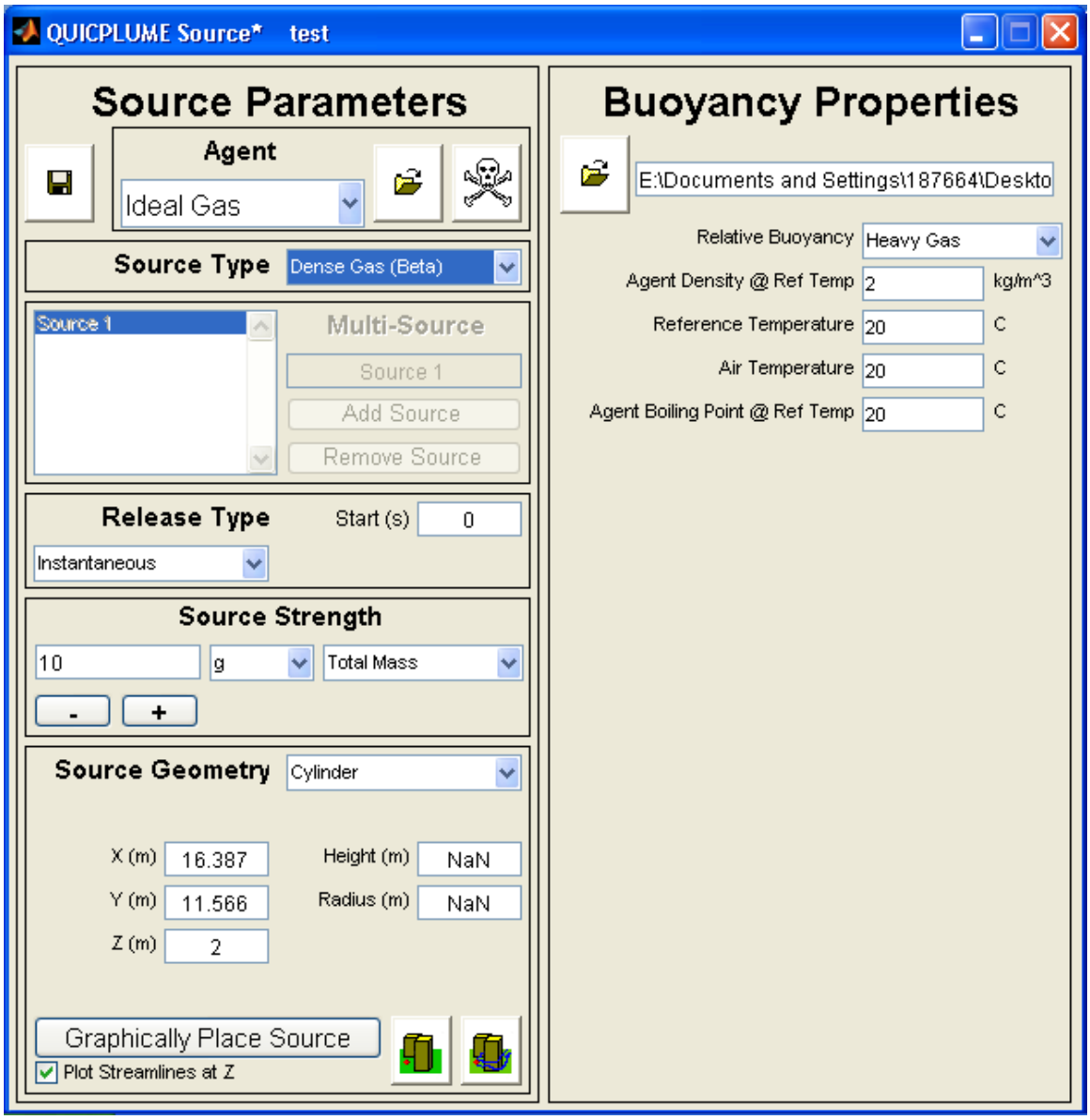

## Source Type

The user can pick from five different source types. They are shown below



# Dense Gas

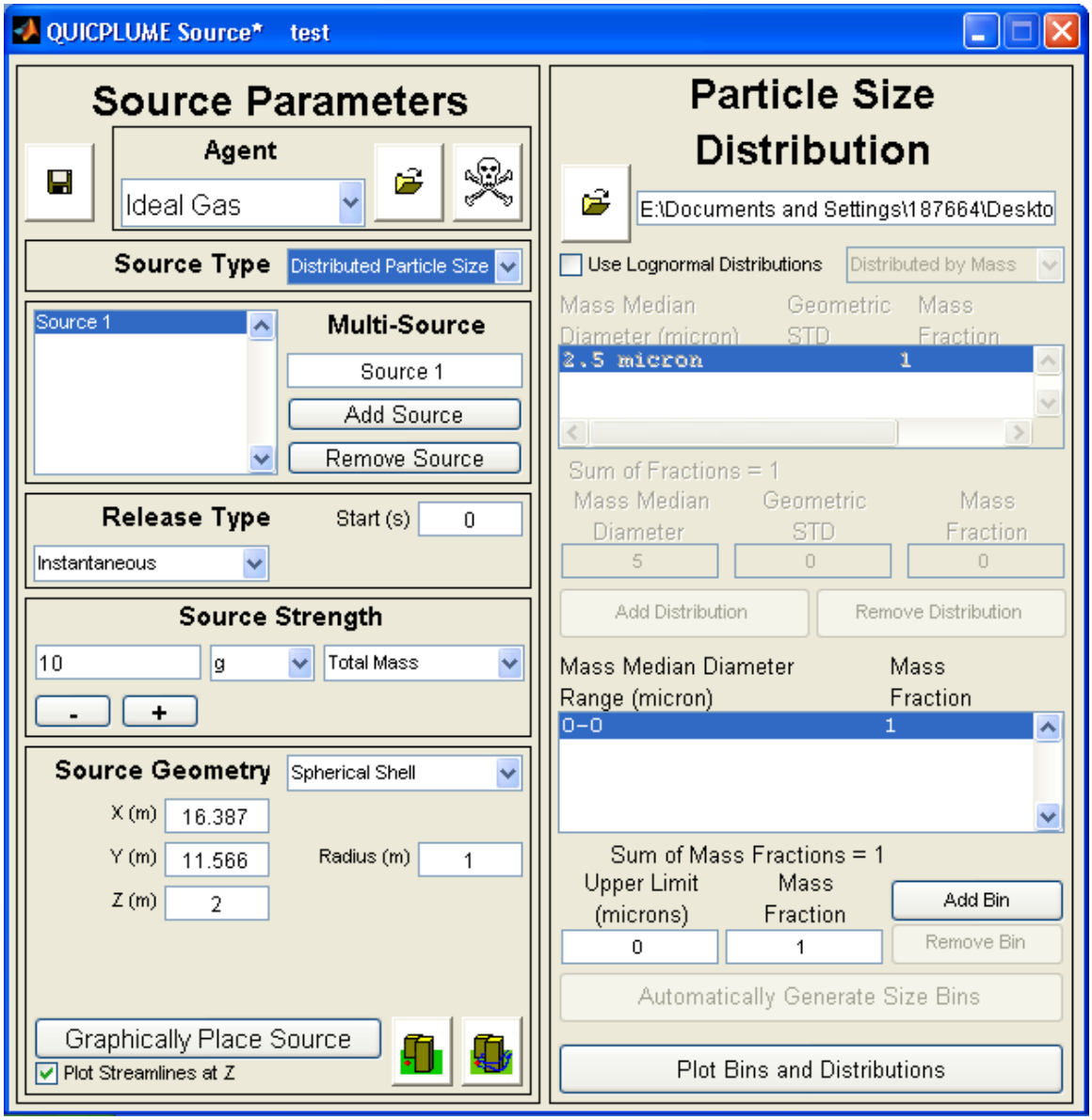
Choosing the 'Dense Gas' option brings up the 'Buoyancy Properties' Window.




The user can manually input values for these properties. In order to choose previously saved values, click the open button. The user is directed to the material library folder, which is the default location buoyancy property files are stored. Alternatively the user may use this GUI to navigate to a project folder where the desired buoyancy properties have been saved previously. After choosing the desired QP_buoy.inp file, click on the open button Open. This will load these values into the 'Buoyancy Properties' window

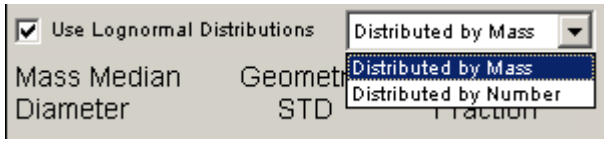
# Distributed Particle Size

Choosing the 'Distributed Particle Size' option brings up the 'Particle Size Distribution' window as shown below.




The user can choose 'Lognormal Distribution' of particle sizes by checking the 'Lognormal Distribution' option as shown below.

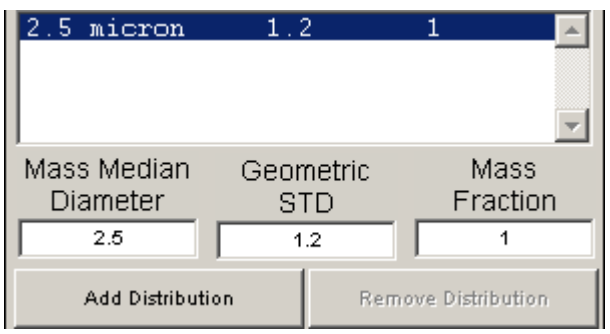
The log normal distributions can be distributed by either mass or by particle count. The distribution type is selected with the pop-up menu next to the checkbox that enables the log normal distribution option.



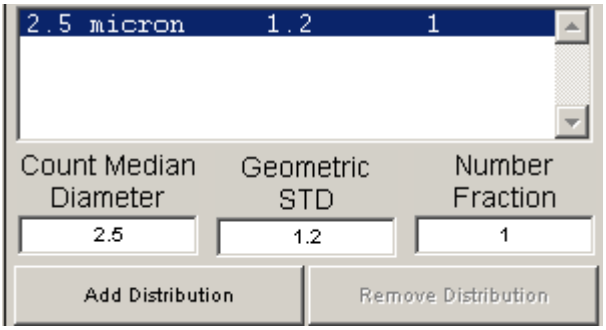

Log normal distributions are defined by a representative diameter (μ), which is the mass median diameter in the case of distributions by mass and count median diameter in the case of distributions by count, and a geometric standard deviation (σ). These parameters define a probability density as a function particle diameter (*d*) shown below:

$$pdf = \frac{1}{d\sigma\sqrt{2\pi}} \exp\left(0.5\left[\frac{\ln(d) - \ln(\mu)}{\ln(\sigma)}\right]^2\right)$$

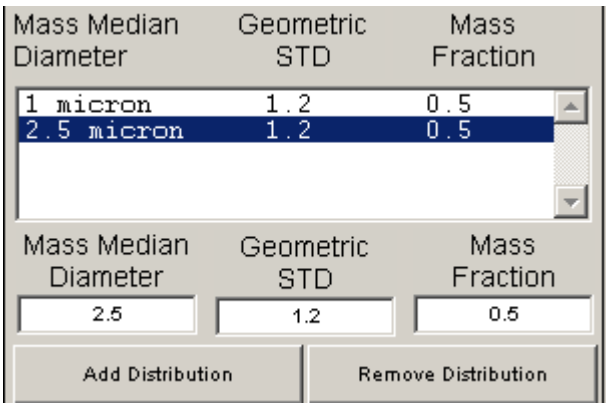
In the case where multiple distributions are used to define the total distribution a mass fraction (for mass distributions) or a number fraction (for count distributions) is also required. If the user chooses the 'Distributed by Mass' option, he / she can enter the values of the mass median diameter, geometric standard deviation and mass fraction in the input box as shown below.

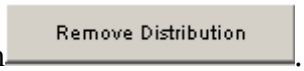



If the user chooses the 'Distributed by Number' option, he / she can enter the values of the count median diameter, geometric standard deviation and number fraction in the input box as shown below.




The user can add distributed particle sizes by clicking the 'Add Distribution' button and entering the values of the mass median diameter, geometric standard deviations etc of the various particle sizes. This is shown below.




To remove any distributions, the user can select the distribution with the cursor and click on the 'Remove Distribution' button 

.

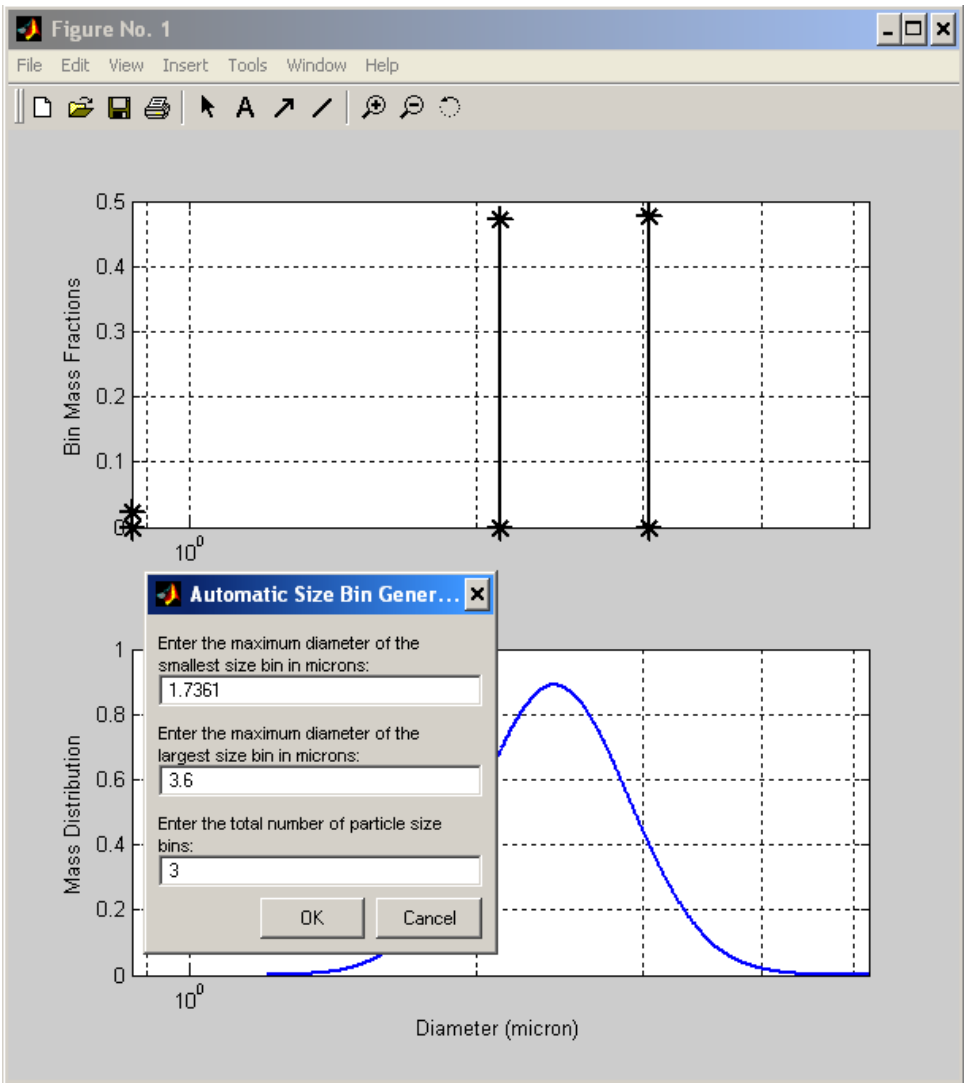
If lognormal particle size distributions option is not selected, QUICPLUME can use a particle size distribution composed of discrete bins with corresponding mass fractions. In case of multiple particle size bins, the lower limit of the second bin (bin with bigger particle sizes) is taken as the upper limit of the first bin (bin with smaller particle sizes). QUICPLUME assumes that all of the particles in a particular bin have the mean bin diameter. The smallest bin has a lower limit of zero yielding a mean size of half the upper limit.

Mean Size of particles in a bin = (Upper Limit + Lower Limit) / 2

For example, if there were two bins with particle sizes distributed between 0-5 micron and 5-10 micron; the mean particle size for the first bin would be (0+5) / 2 = 2.5 micron; the mean particle size for the second bin would be (5+10)/2 = 7.5 micron (upper limit of first bin = lower limit of the second bin).

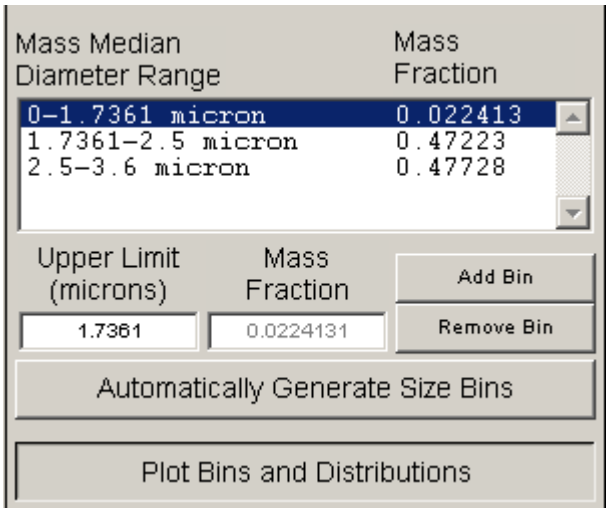
The ‘Particle Size Distribution’ window provides the user the option of either entering these bins manually using the ‘Add Bin’ button Add Bin or they can use the ‘Automatically Generate Size Bins’ option Automatically Generate Size Bins to produce discrete size bins from the lognormal distributions.

Note: This function was originally introduced when the discrete size bins were the only method to input particle size distributions. Now that QUIC-PLUME uses the lognormal distributions to produce continuous particle size distributions directly, this functionality is obsolete.



In the ‘Automatic Size Bin Generator’ window, enter the values as shown and press the ‘ok’ button. This leaves the user with a figure showing the ‘Bin Mass Fraction’ and ‘Mass Distribution’ with respect to the bin diameter.

Closing this figure takes the user back to the main ‘Particle Size Distribution’ window and here the user can see the bins generated and their corresponding mass fractions.

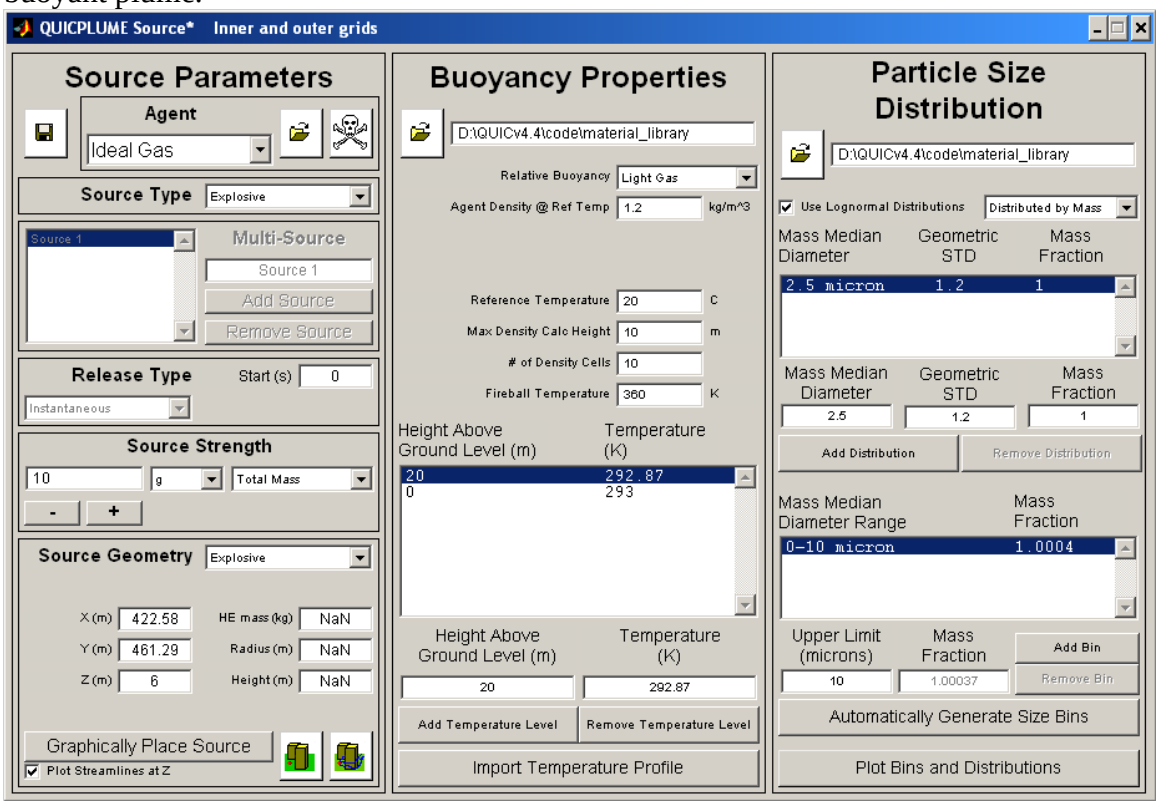



To view the distribution of particle sizes within each bin, click on the ‘Plot Bins and Distributions’ button. and this brings up the same figure that shows the ‘Bin Mass Fraction’ and ‘Mass Distribution’ plotted with respect to the bin diameter.

In order to choose previously saved particle distributions, click the open button. The user is directed to the material library from where he / she can choose a previously saved QP_particlesize.inp file or the user may navigate to a previously saved project and load the QP_particlesize.inp file from another project folder. After choosing the desired file, click on the open button. This will load the new values into the ‘Particle Size Distribution’ window.

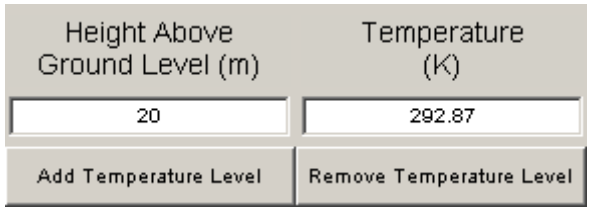
# Explosive

Choosing the 'Explosive' option brings up the 'Buoyancy Properties' and 'Particle Size Distribution' window as shown below. Using this option QUIC-PLUME computes the buoyant rise of the plume internally using the initial explosion characteristics and the thermal properties of the atmosphere entered in the Buoyancy Properties panel. This source type is recommended over the ERAD source type, which also simulates explosive releases, because this method takes into account the effects of the local winds on the buoyant plume.

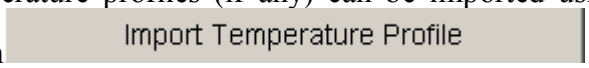



Enter the Buoyancy properties of the explosive gas in the buoyancy properties window.

Temperature levels can entered manually by typing in the values in the 'Height Above Ground Level' and 'Temperature' boxes and clicking the 'Add Temperature' button



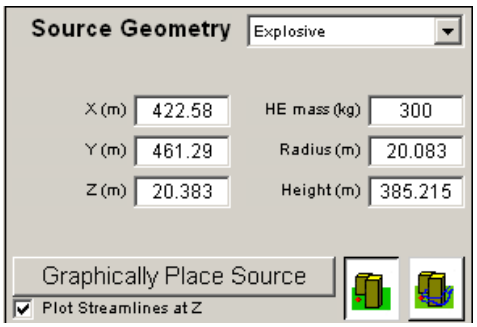

Existing temperature profiles (if any) can be imported using the 'Import Temperature Profile' button



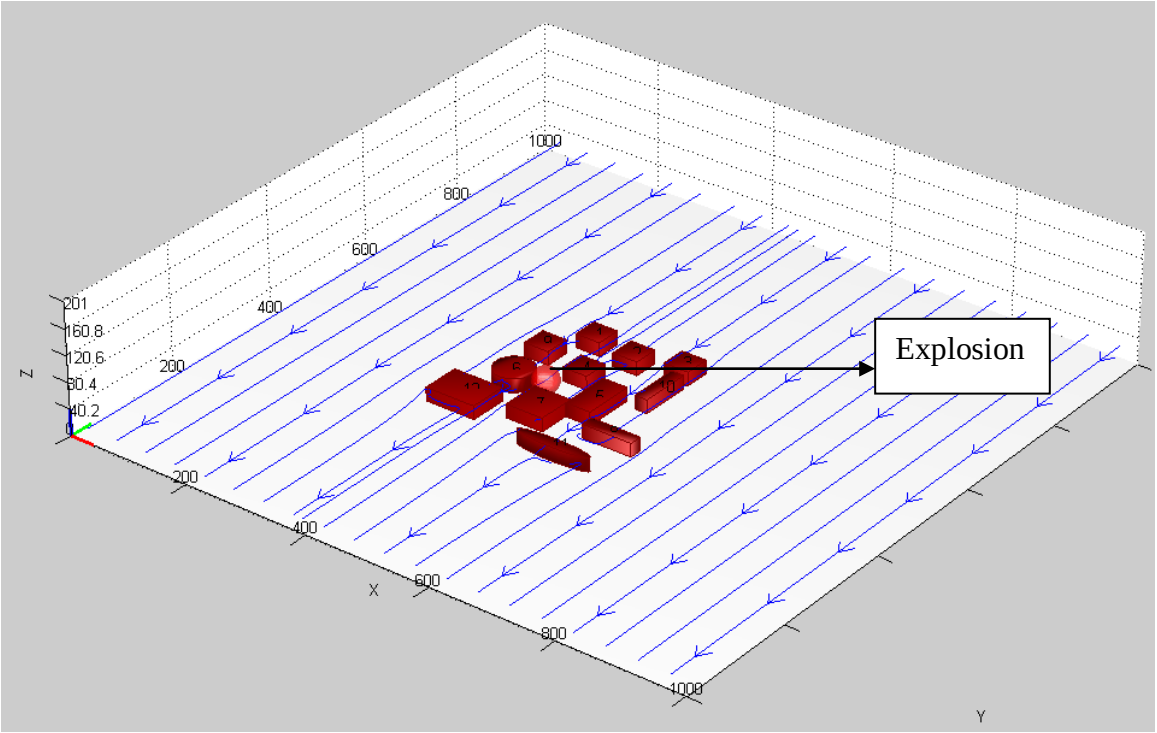
Enter the values in the 'Particle Size Distribution' window based on the explanation given in the previous section.

After performing these steps, move to the 'Source geometry' portion of the main window. Here the user will find the source geometry as 'Explosive'.

Enter the locations and mass of the high explosive (HE) in kg. The initial blast radius (one atmosphere overpressure radius) and approximate height of the plume after it comes to equilibrium are calculated based on the amount of HE mass. This is shown below.



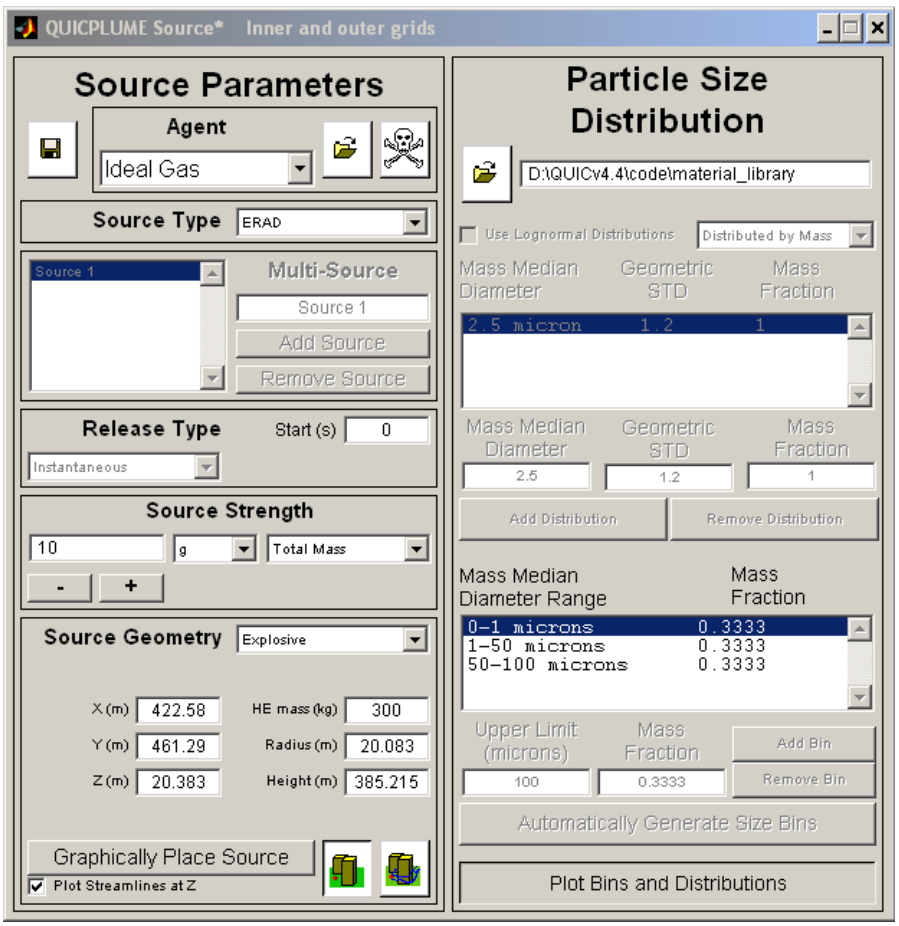

To view the location of the explosive in the city, the user can use the 'View source placement in the city' button .



# ERAD

Choosing the 'ERAD' option brings up the 'Particle Size Distribution' window as shown below. The ERAD source type also simulates an explosive release similar to the Explosive source type. The principal difference lies in the fact that for this source the particle distributions and equilibrium plume are initialized at the source location using output from the ERAD model. Thus, it is assumed that the time that it takes for the buoyant plume to come to equilibrium is negligible and the distribution of the various sizes of particles in the plume occurs instantaneously.

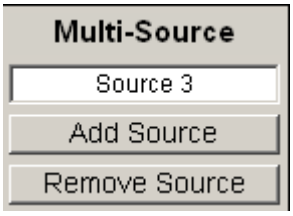
The user can import previously saved parameters by clicking the 'open' button and navigating to the desired QP_erad.inp file from the material library folder or he / she can use any other files by navigating to their respective folders.



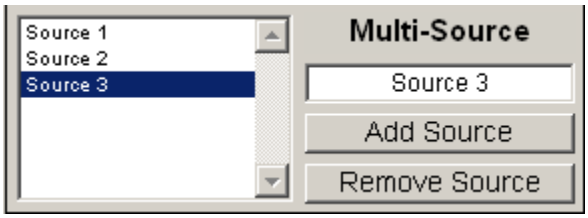

Under the 'Source Geometry' section, enter the location and mass in kilograms. To view its location in the city, the user can use the 'View source placement in the city' button .

## Multi-Source

Some source types (basic and distributed particle size) can have multiple sources in a single simulation. The user can choose multiple sources by clicking on the 'Add Source' Add Source button under 'Multi-Source'



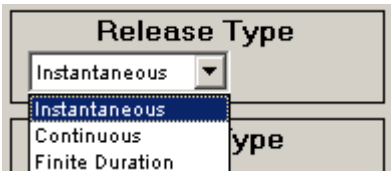

For each of the sources added, the user can input different source parameters by selecting the source and entering the values of the various parameters



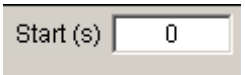

To remove a source, the user can select the source with the cursor and click on the 'Remove Source' button Remove Source

## Release Type

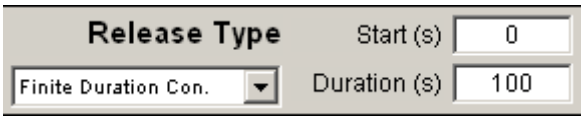
Three different release types can be selected from the pull-down menu: instantaneous, continuous, and finite duration as shown below. 'Instantaneous' means that all of the particles are released at once. 'Continuous' means that the particles are released at a constant rate from the source start time through the entire duration of the simulation. 'Finite Duration' means that the particles are released at a constant rate for the amount of time entered in the text box that appears next to the release type pull-down menu when finite duration is selected.



The user can specify the start time for the release by entering the time in the 'Start (s)' window as shown below. The default value for the 'instantaneous' and 'Continuous' type releases is set to zero.

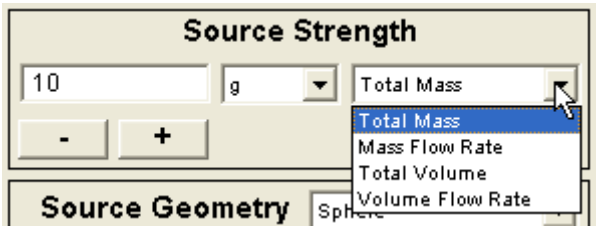



For the 'Finite Duration' type release, the user should specify the start time and duration of the release as shown below.

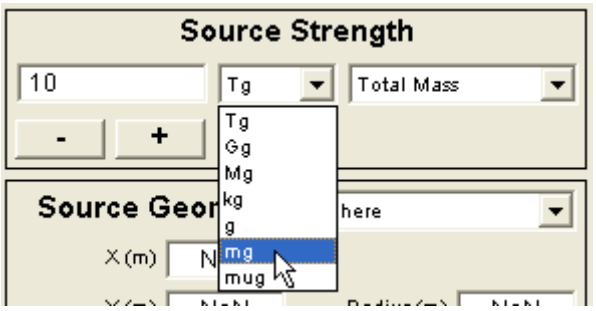


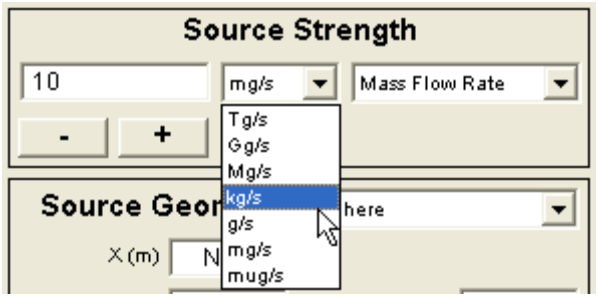

## Source Strength

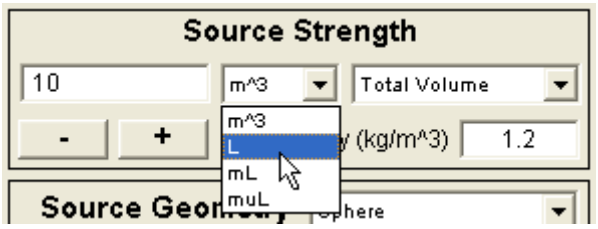
The size of the release is controlled through the "Source Strength" text box. The amount released can be represented as total mass, mass flow rate, total volume and volume flow rate as shown below.

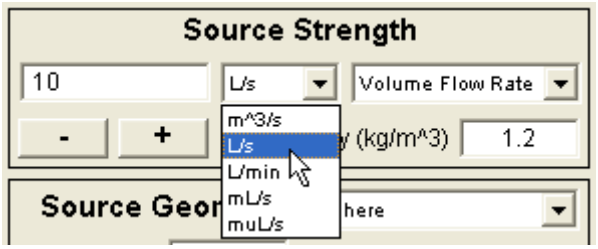



Choosing the source strength type determines the units available in the Source Strength Units Pull-Down Menu as is shown below. Since QUIC-PLUME always uses mass for material amounts, volumetric source types also require a fluid density in kg/m$^3$.





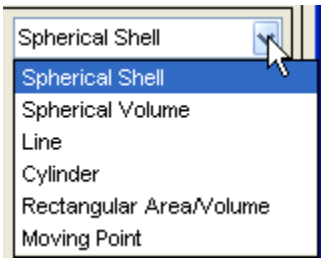




After selecting the source strength type and units the user can either use the edit box to edit the source strength value or increase or decrease the value using the increment – decrement buttons

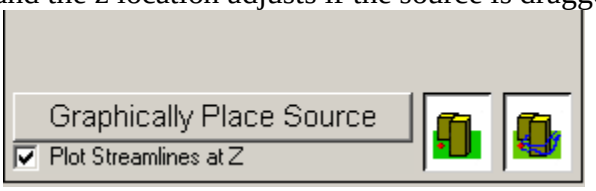
## Source Geometry and Placement

Six source geometries are available to the user in QUIC-PLUME. They are sphere, line, cylindrical area & volume, rectangular area & volume, explosive, and moving point sources. The source type selected determines the geometries that are available for use in a project. Currently the dense gas source is uses the cylindrical area/volume source and the explosive source geometry is only used in the explosive and ERAD source types. The geometries that are available for basic and distributed particle size source types are shown in the figure below.



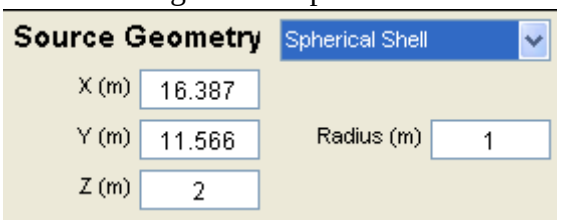
Choosing 'Spherical Shell' or 'Spherical Volume' creates a sphere centered at the location specified in the X, Y, Z location text boxes with the given sphere radius. The shell places all of the particles at the radius specified while the volume distributes the particles evenly throughout the sphere. 'Line' creates a straight line between the points defined in the X, Y, Z begin and end point text boxes. Note that the line can have multiple segments, choosing the segment updates the beginning and end point edit boxes with the values for the selected line segment. Choosing 'Cylinder' creates a cylinder centered at the location specified in the X, Y text boxes with the given height and radius. Note that choosing a height of zero creates a circular area source. Choosing 'Rectangular area / volume' creates a volume with the specified length, width and height. Note that choosing one of the dimensions as zero can result in a 2D plane area source. Choosing 'Moving Point' allows one to create a path between user-specified waypoints and to specify a velocity of the point at each way point. The source will have a constant acceleration between way points determined by the beginning and ending velocities. Particles are randomly placed on a line, along the traveled path over a time step, in a plane, or within volumes when released.

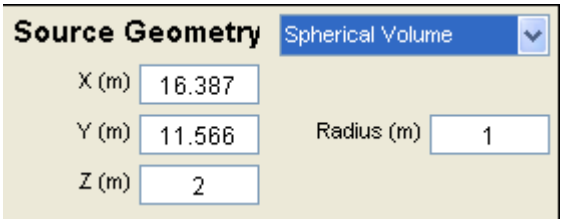
The source location can be set using the text boxes or by clicking 'Graphically Place Source'. Notice that as you graphically move the source around, the X and Y locations update automatically and the z location adjusts if the source is dragged over a building.



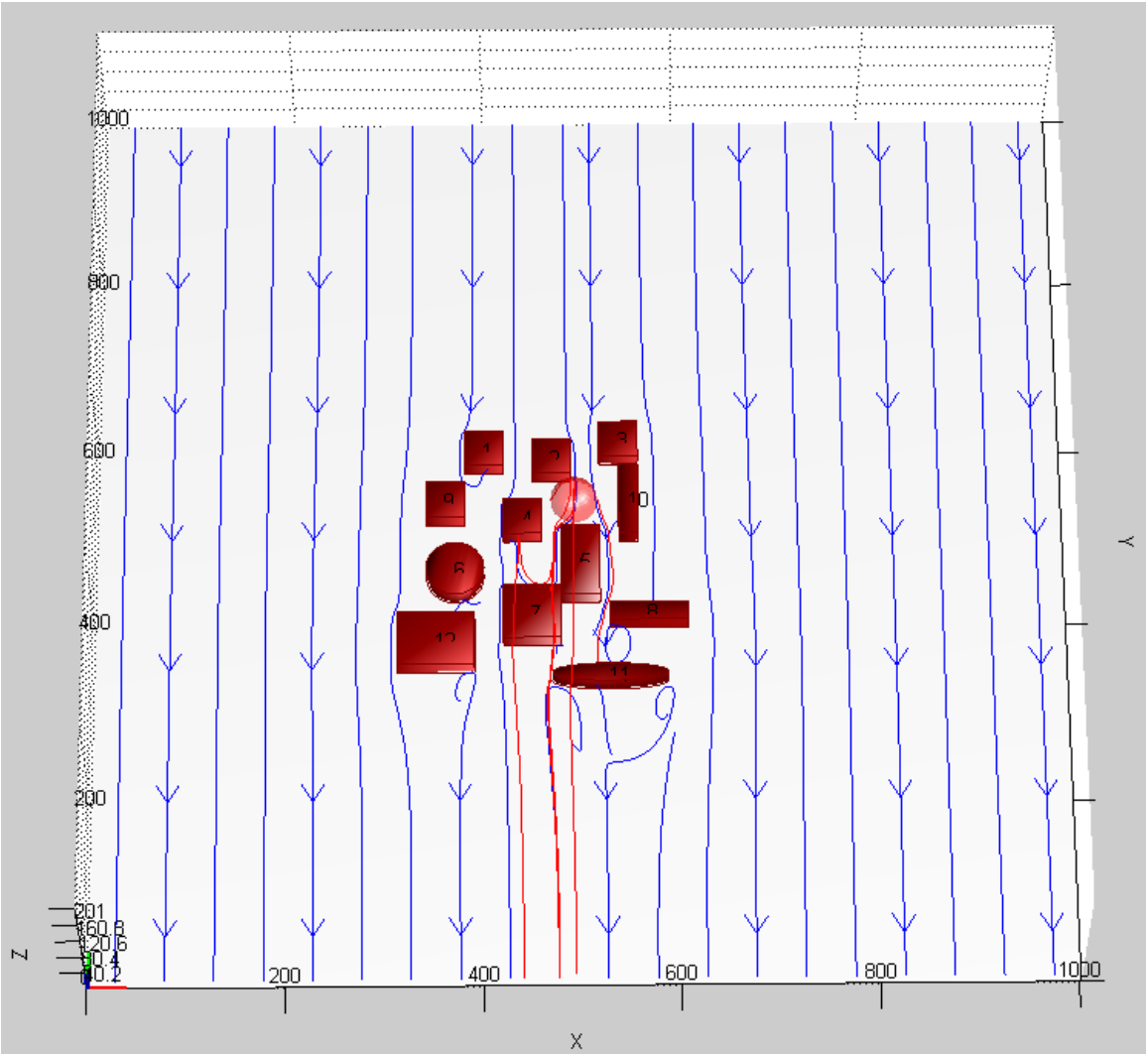

## Spherical Shell and Volume

To place a spherical source where all of the particles are placed at the radius, select the source geometry as 'spherical shell' and enter the co-ordinates of the location and radius of the sphere as shown below. Selecting 'spherical volume' evenly distributes the particles throughout the spherical volume.




To view the location of the source, press either the 'View the source placement within the city' button or 'View the source emitted streamlines button'

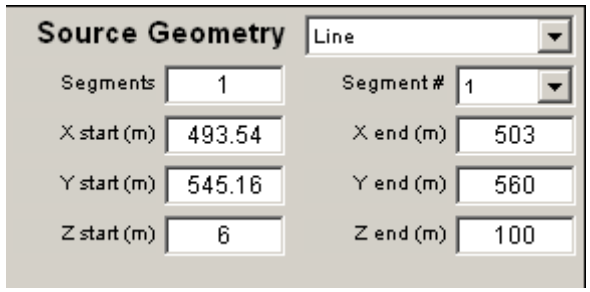


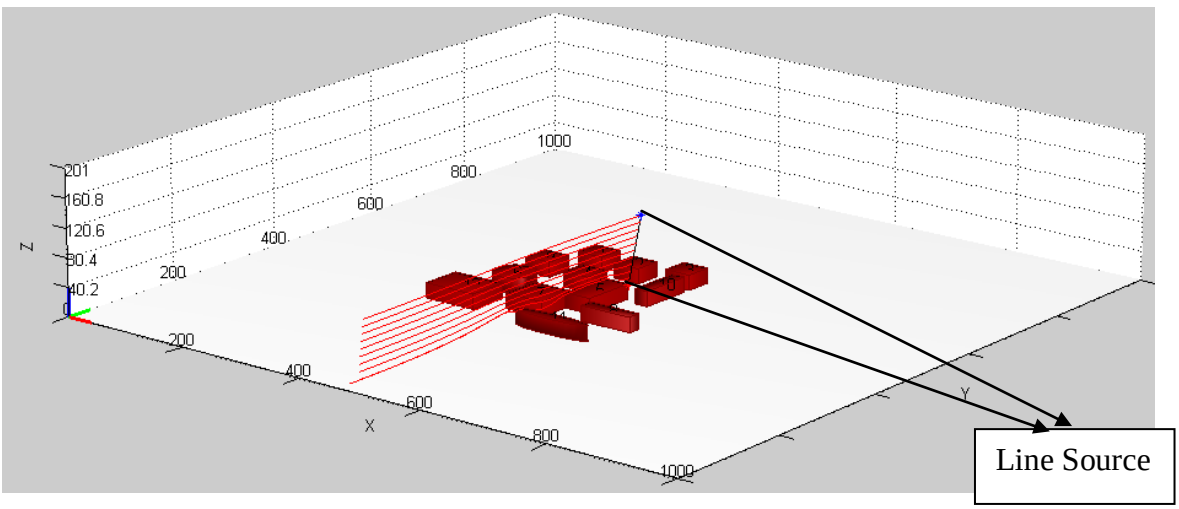

To graphically place the source, click the 'graphically Place Source' button Graphically Place Source

This will launch a 'Plume Source Placement' window shown below. With the cursor, place the source at the desired location. The 3D image manipulation buttons at the top of the window can be used to zoom in on a particular area of the city if greater detail is required for graphical source placement.

# Line

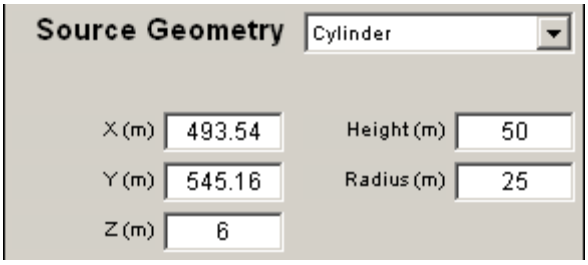
To place a line source using the text box option, select the source geometry as 'line' and enter the co-ordinates of the start and end locations of the line as shown below. Alternatively, this could be done using the 'Graphically Place Source' option. Graphically defining a line source involves left-clicking the number of segments plus one locations on the Plume Source Placement window to define the beginning and end of all the segments.

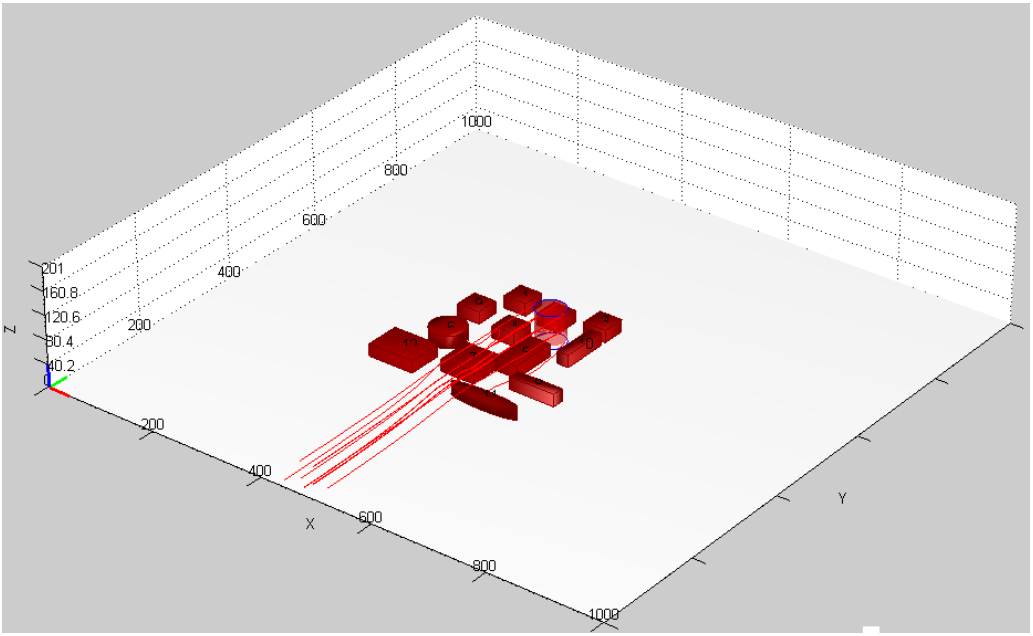


# Cylindrical Area/Volume

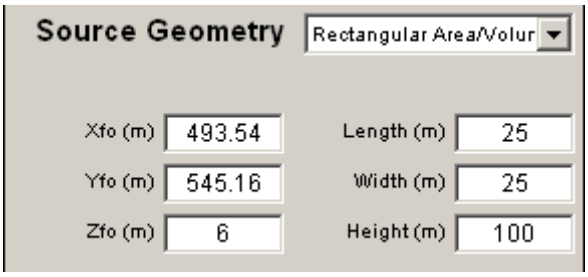
To place a cylindrical source using the text box option, select the source geometry as 'cylinder' and enter the co-ordinates of the location of the cylinder and also the height and radius of the cylinder. Alternatively, this could be done using the 'Graphically Place Source' option.

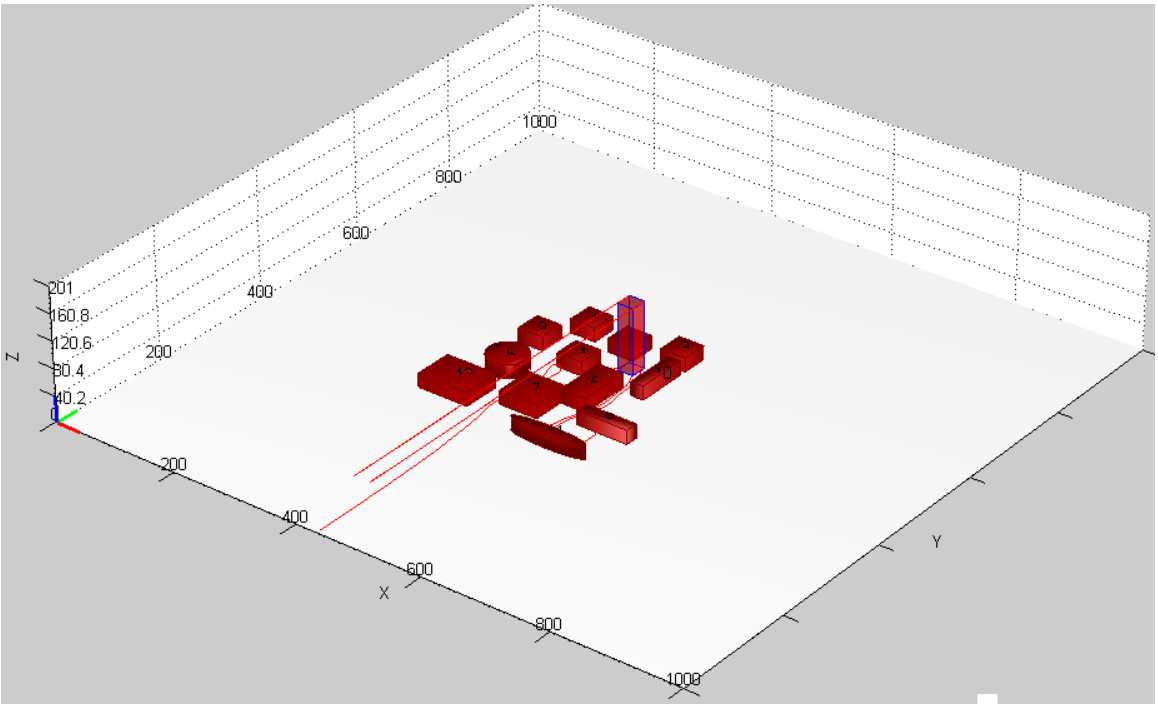


# Rectangular Area / Volume

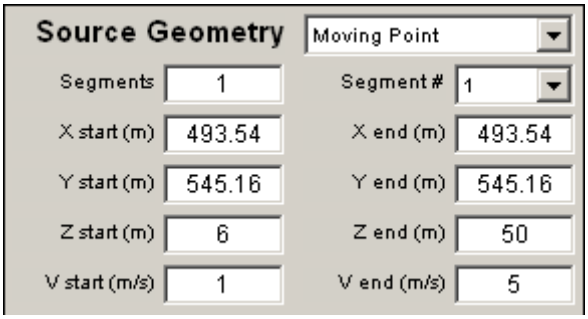
To place a rectangular source using the text box option, select the source geometry as 'Rectangular Area / Volume' and enter the co-ordinates of the location of the source and also length, breadth and height (if the source is a volume). Alternatively, this could be done using the 'Graphically Place Source' option.

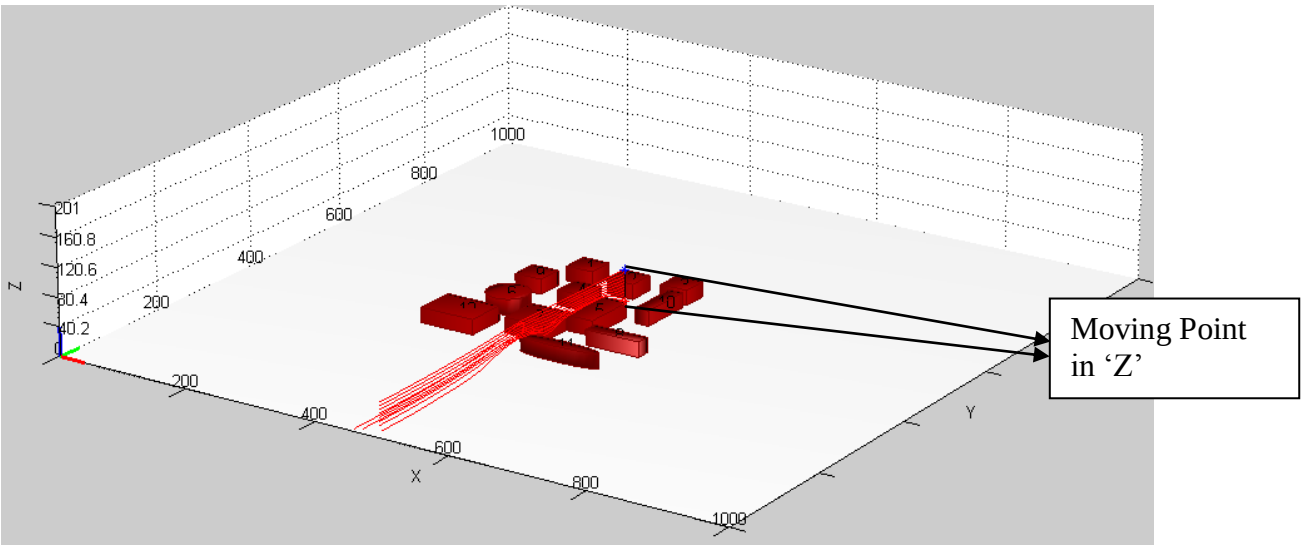


# Moving Point

To place a moving point source using the text box option, select the source geometry as 'Moving Point' and enter the co-ordinates of the start and end locations of the source and also velocity of the source at the start and end locations. Alternatively, this could be done using the 'Graphically Place Source' option.



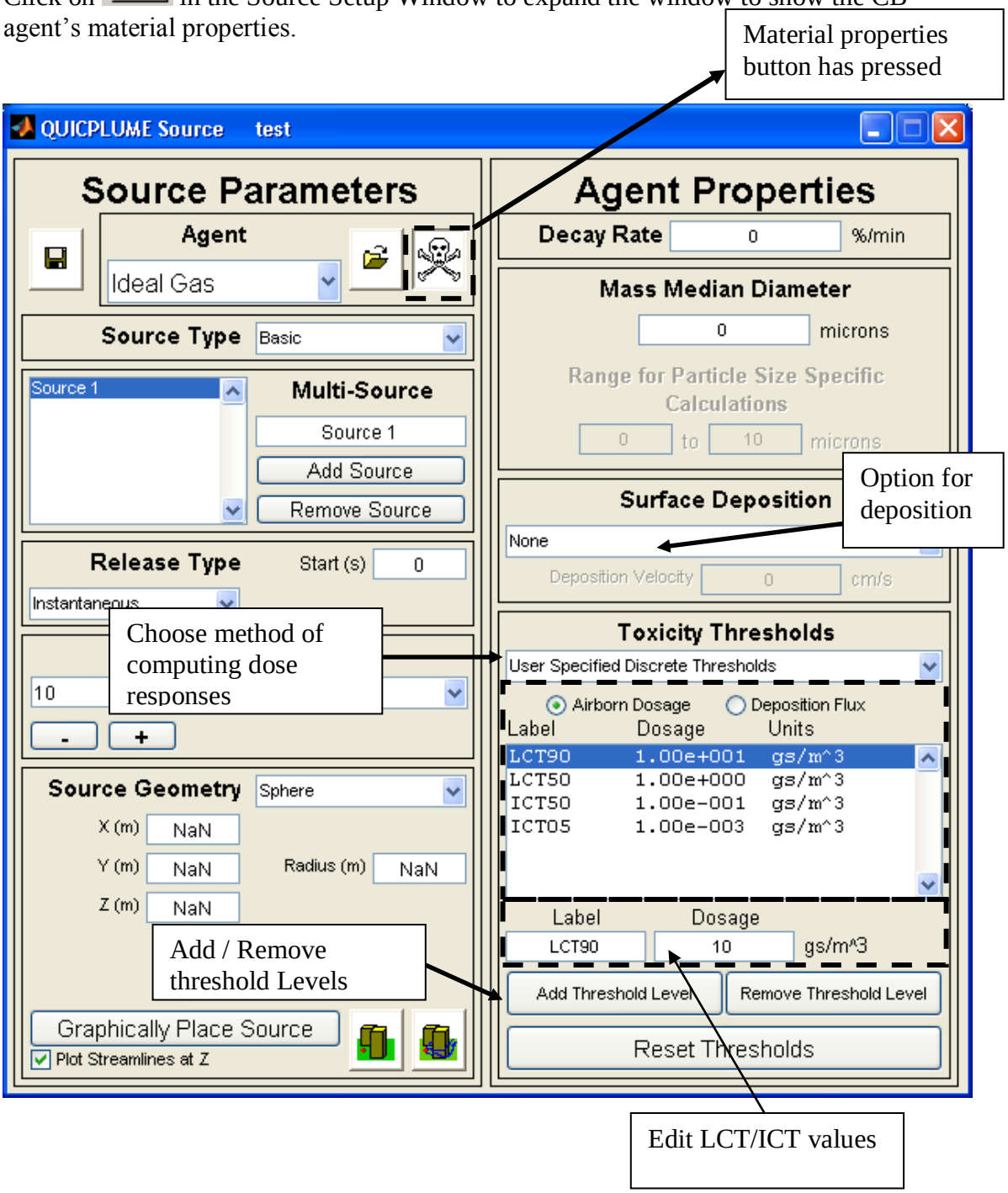




Note: There are checks in the graphical source placement functions to prevent the placement of any part of the source inside of a building and the vertical position of the source is automatically selected to avoid placing any part of the source inside of a building. This can be overridden through the text boxes but this should be avoided as it will cause the QUIC-PLUME code to crash.

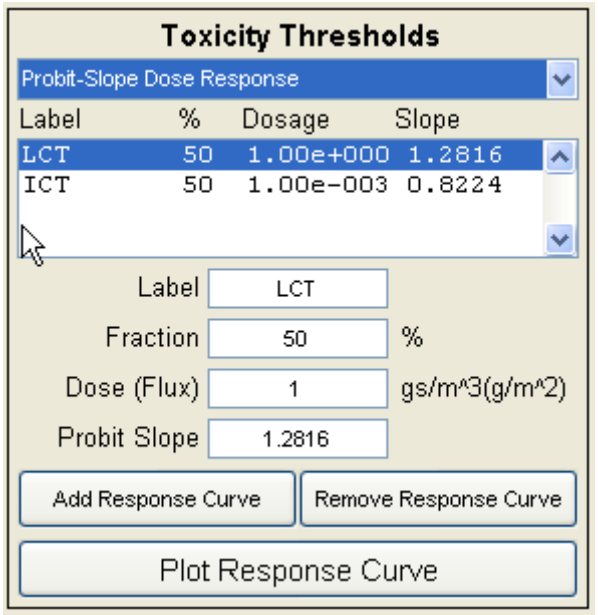
You will need to click on to save current parameters and close the Source Placement Window.

## Basic Agent Properties

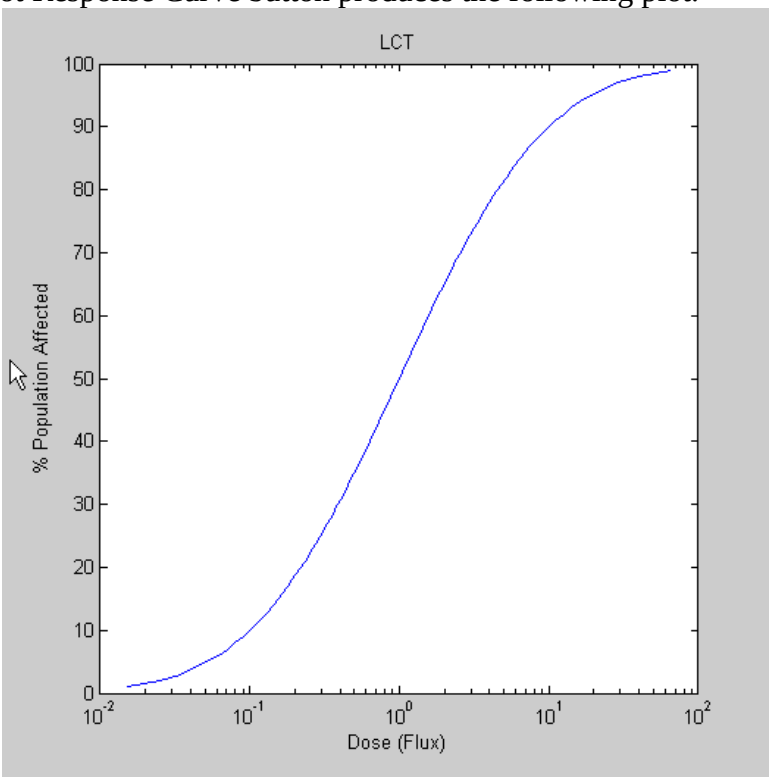
Click on in the Source Setup Window to expand the window to show the CB agent's material properties.



Dose response contours can be produced by two different methods: user input discrete levels the probit slope method which produces continuous contours of dose responses. The figure above showed the controls when the discrete method is selected. When the probit slope method has been selected the following controls appear in the place of the discrete threshold controls.




Pressing the Plot Response Curve button produces the following plot.



## Agent Library

One can define the following agent properties: decay rate, mass median diameter, deposition velocity, and LCT (lethal concentration threshold) values. Default values appear based on the agent type selected in the Agent Type pull-down menu to the left. These values are found in a text file called materials.lib. The public release version contains a library file with generic agents (ideal gas, 5 micron particles, etc.) that have artificial material property values. An unclassified official use only agent library is provided upon request to government and other qualified users.

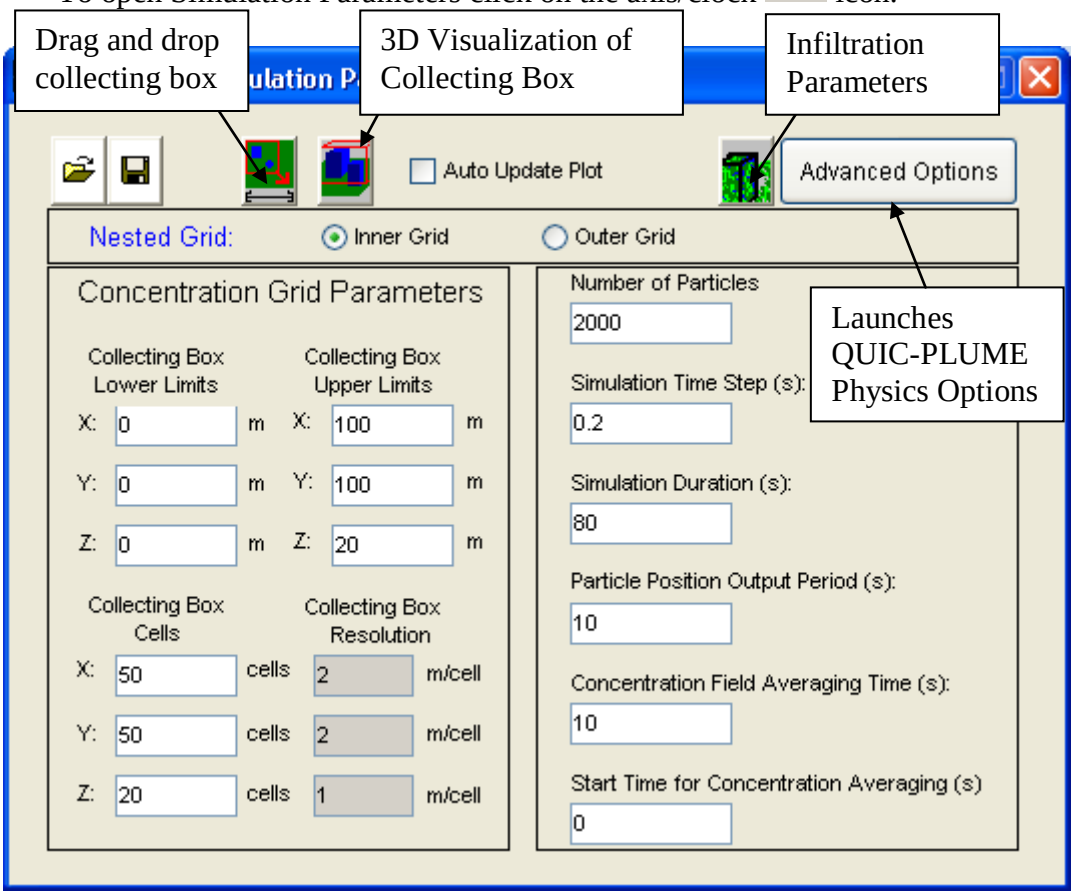
New agents can be added to the library. Simply append the new agent's material properties to the end of the materials.ref file in the code folder. The format for the material agent entries is comma delimited in the following order: material name, decay rate (%/min), mean particle diameter (micron), deposition velocity (cm/s)[a value of -99 indicates internally calculated deposition velocity and a value of 0 indicates no deposition], number of probit slope response curves, the first probit label string, the first probit fraction, the first probit dose (g*s/m^3 or g/m^2), the first probit slope, second probit label string (if any),…, the first LCT/ICT level value (g*s/m^3), first LCT/ICT level label, second LCT/ICT level value (if any), … An example of a agent entry is shown below:

Ideal Gas,0.,0.,0.,2,LCT,50,1,1.2816,ICT,50,1e-3,0.8224,10,lct90,1.0,lct50,1.0e-01, ict50,1.0e-03,ict05

After making any changes in the Source Parameter Set-up Window, you will need to click on to save current parameters and close the Source Placement Window

# QUIC-PLUME Simulation Parameters

- To open Simulation Parameters click on the axis/clock icon.




- The set-up of the concentration domain is done on the left, while several QUIC-PLUME random-walk parameters, averaging time parameters, and output frequency parameters are specified on the right. Advanced options (such as atmospheric stability, particle splitting, etc.) are defined in the QUIC-PLUME Physics Options GUI which is launched by the 'Advanced Options' button in the upper-right-hand corner.

- The 3D extent of the concentration grid can be set through the "Collecting Box Lower and Upper Limits". By default, the horizontal extent of the domain is the QUIC-URB modeling domain minus a one grid cell skirt.

- Specification of the number of grid cells through the "Collecting Box Dimensions" text boxes controls the grid size. The grid cell can be any size and does not need to be equivalent to the grid cell size of the QUIC-URB domain.

- There is a tradeoff in changing the number of particles in the random-walk simulation: more particles results in a computed concentration with less uncertainty, but fewer particles results in a faster simulation. The higher uncertainty produced by fewer particles may be compensated by larger collecting box grid cells or longer averaging times. Note that for finite duration and continuous releases, the QUIC-GUI will automatically round-off the number of particles you enter to satisfy the constraint that an integer number of particles must be released at each time step.

- The simulation time step controls the size of the time step used by the random walk model. A larger time step allows the simulation to proceed more quickly, but the time step must be smaller than the Lagrangian time scale to give accurate answers. The code will internally compute time step constraints and change the time step when necessary. However, choosing an "unreasonable" time step, either way too large or way too small, will lead to problems. For outdoor, real-scale building-resolved problems a time step of a few seconds is reasonable. For wind tunnel scale problems, a tenth of a second time step may be required. Note: a simple approximation for the Lagrangian time scale over a flat surface is kz/u*, where k is the von Karman constant, z is height above ground, and u* is the friction velocity.

- The "duration of the simulation" controls how long the dispersion calculation will proceed. If a finite duration release has been selected, the simulation duration must be at least as long as the release duration.

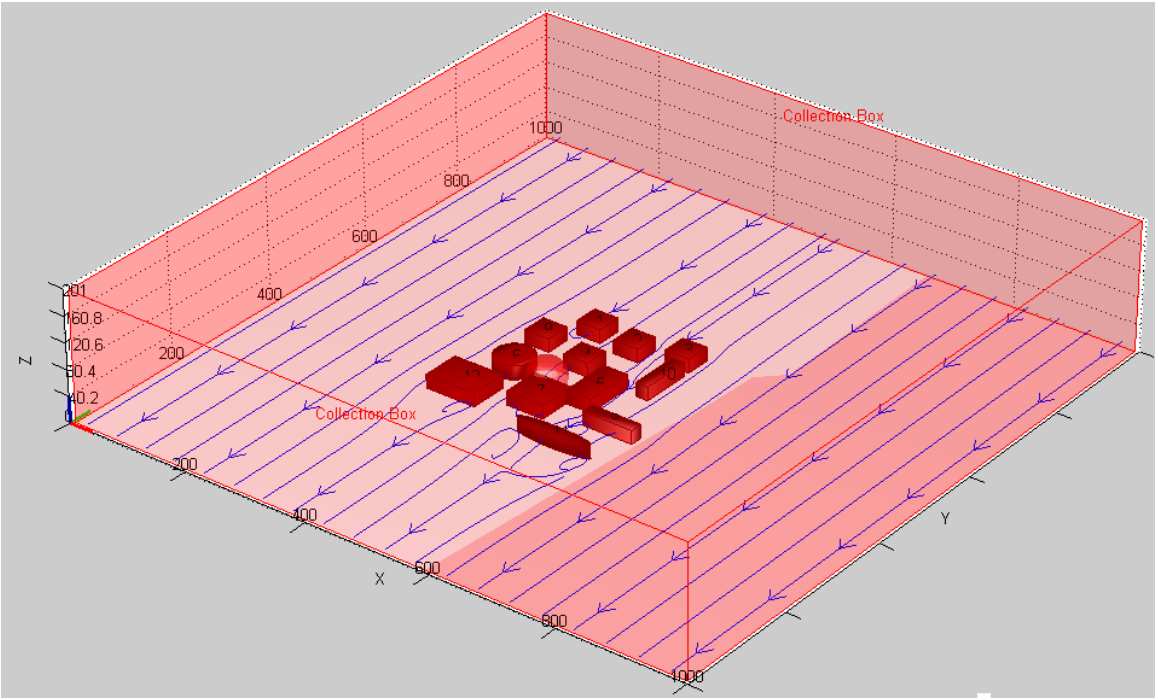
- The particle output frequency determines how often particle location data is written out to a text file (which is then used in the VIS-GUI to visually illustrate the dispersal of the agent). This parameter does not affect the physics of the random-walk model, it only affects how often particle positions are written out to a file. When running simulations with large numbers of particles, it is strongly recommended to set this to a large number in order to save disk space (and speed up the simulation). When making animations of particle transport and dispersion, it is recommended that a small number of particles (e.g., 500 to 2000) be used along with a small number for the output frequency (e.g., every time step).

- The QUIC-PLUME code produces time-averaged concentrations in a 3D volume defined by the concentration collecting box. The "Concentration Field Averaging Time" specifies the time interval over which particles are counted in the collecting grid boxes.

- The "Concentration Average Start Time" specifies when the concentration averaging will begin. This is effectively a skip time at the beginning of the particle simulation over which the concentration is not computed. By default it is set to 0.

## View Current Collecting Box

The ‘View Current Collecting Box’ option helps the user visualize the concentration domain of QUIC-PLUME. This is shown below.

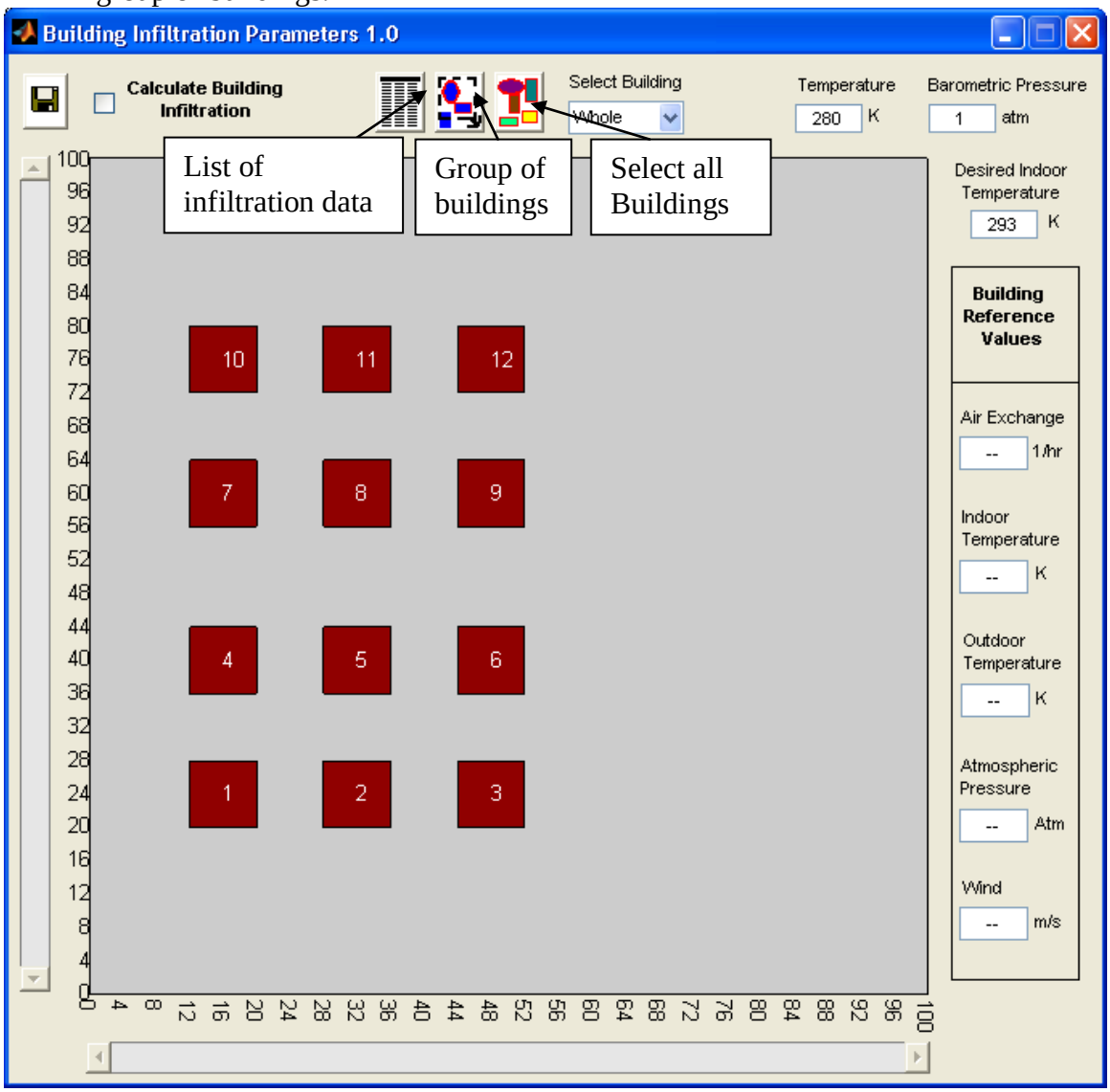


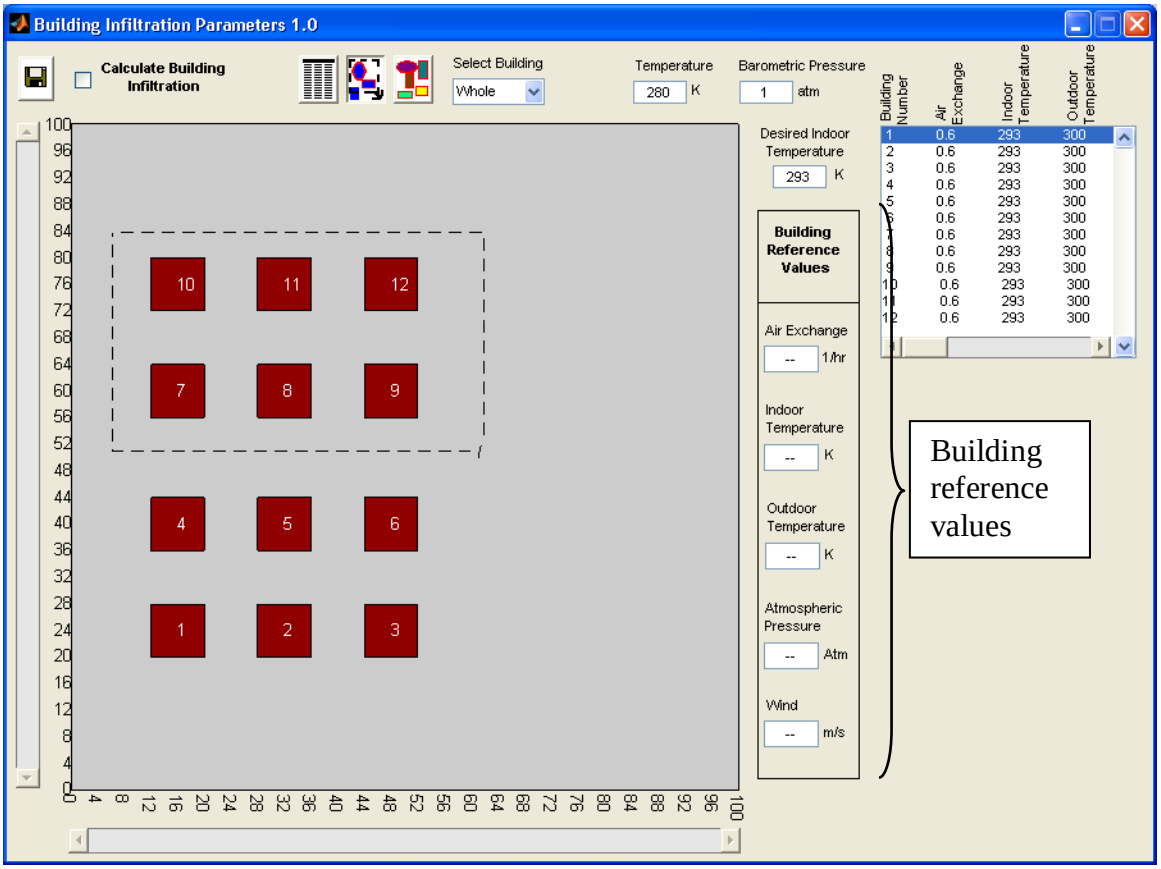
## Edit Collecting Box

The 'Edit Collecting Box' provides the user the option of editing the bounds of the concentration domain. To do this, click on the 'Edit Collecting Box' button and with the cursor, change the limits of the domain. This is shown below.

## Building Infiltration

- Pressing the 'Building Infiltration Parameters' button opens up a window where the building infiltration parameters can be set. Warning: this is a beta feature and has not been fully tested.
- By default, the calculation of building infiltration is turned off in QUIC. In this window, you can turn it on by checking the "Calculate Building Infiltration" checkbox.
- Selecting the 'List of Infiltration Data' shows the infiltration data associated with each building in table format. Currently this feature does not work correctly.
- To select all the buildings in the domain, choose the 'Select all buildings' option
- To select a particular building or a portion of a building, choose the 'Group of Buildings' option. One can then drag out a region with the mouse and select a group of buildings.





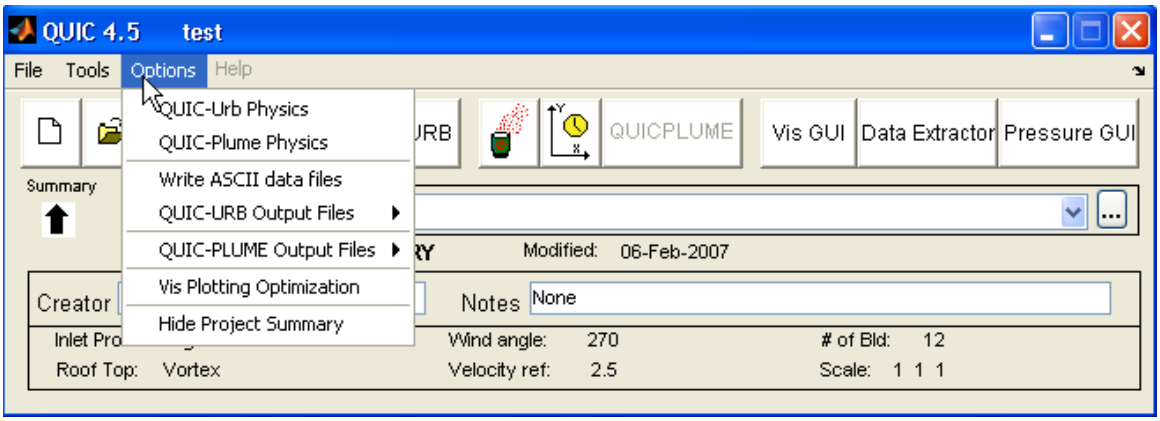

- Once a building or group of buildings is selected, then one can enter building reference values, such as air exchange rate, the indoor & outdoor temperature, the wind speed, and the barometric pressure in the boxes to the right. These are called reference values, because the air exchange rate is specific to a reference case for specific indoor and outdoor temperatures, wind speed, etc. The temperature, barometric pressure, and the desired indoor temperature are the current values, the values for the current simulation. QUIC will take these values, along with the wind speed computed by QUIC-URB, and then modify the air exchange rate based on the differences between the reference values and the current values.

- Click on to save current parameters before closing window. This brings the user back to the main QUIC-PLUME Simulation Parameters Window.

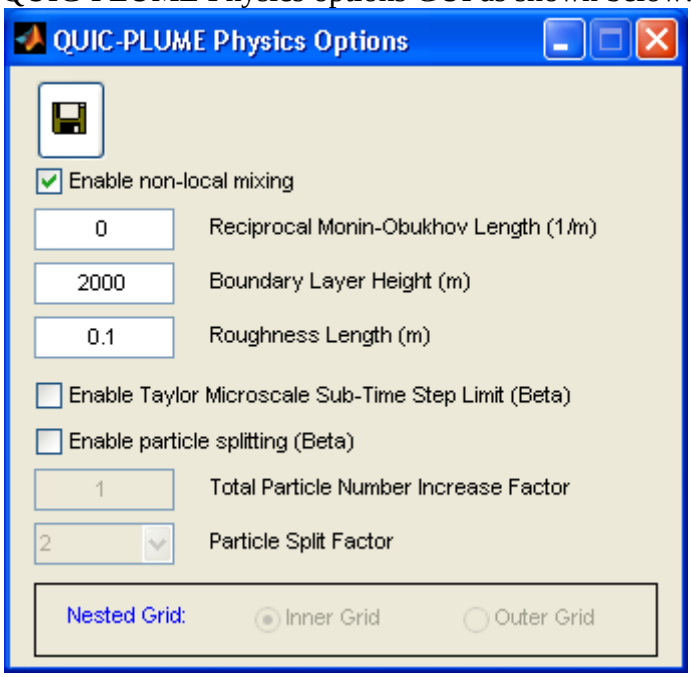
- Click on to save all the set parameters before closing the main window.

# QUIC-PLUME Physics Options

Several parameters used by the QUIC-PLUME dispersion model can be changed through the Physics Options pop-up window. On the main QUIC-GUI window select <Options> <QUIC-PLUME Physics>.




This launches the QUIC-PLUME Physics options GUI as shown below:




## Non-Local Mixing

The non-local mixing option simulates the enhanced dispersion due to the transient large scale vortical structures found in regions such as the cavity region of the building wake. This option is turned on by default.

## Reciprocal Monin-Obukhov Length

Atmospheric stability is entered as a reciprocal Monin-Obukhov length. This will affect the turbulence and consequently the dispersion of the plume. A value of 0 indicates neutral atmospheric stability, the default value which corresponds to an idealized condition typical of wind tunnels or an atmospheric boundary layer under cloudy conditions with at least moderate winds. A positive value indicates thermally stable conditions which is typically associated with the nocturnal atmospheric boundary layer. A negative value indicates thermally unstable conditions which are typically associated with the convective conditions that occur in the atmosphere during the day.

## Boundary Layer Height

The turbulence in QUIC-PLUME decays linearly with height to zero at the boundary layer height. This effect is typically only important in nested grid problems where the outer domain has a very large vertical extent. The default value is 2000 m. This effect can be made negligible by setting the boundary layer height much larger than the vertical extent of the domain.

## Roughness Length

Due to the course resolution of the mean wind fields produced by QUIC-URB it is necessary to interpolate the velocity and turbulence values when particles are close to surfaces. This is done assuming a logarithmic profile defined by the surface roughness length. Particles are also reflected off of the roughness length (i.e., particles are not allowed to be less than the roughness length from any surface). The default value is 0.1 m for atmospheric domains.

## Taylor Microscale (Beta)

QUIC-PLUME determines the Lagrangian time scale for each particle and splits the time step into sub-time steps that are at most half of the Lagrangian time scale. The Lagrangian time scale approaches zero as the particle approaches a surface. The Taylor microscale option puts a lower limit on the size of the sub-time steps. This option is experimental and is turned off by default.

## Particle Splitting (Beta)

If particle splitting is enabled the particles are split when they enter an empty concentration collecting box. Two parameters are needed when using the particle splitting option: total particle increase factor and the particle split factor. The total particle increase factor multiplies the total number of particles indicated in QUIC-PLUME Simulation Parameters GUI to determine the total number of particles allowed in a simulation. Particles will be allowed to split until the total number of particles reaches this number times the original number of particles in the simulation. The particle split factor determines the number of particles a particle will split into when it meets the conditions of splitting.

# Running QUIC-PLUME

Once the QUIC-PLUME source and simulation parameters have been defined and saved, the QUIC-PLUME code can be run (if the parameters are not specified, the QUIC-PLUME button will be deactivated).

- Select the "QUICPLUME" icon.


The light bulb indicates that QUIC-PLUME is running and the wind fields are being computed.

When the light bulb disappears, the model simulation is finished. The output of particle and concentration fields can then be visualized with the Vis-GUI (see next section).

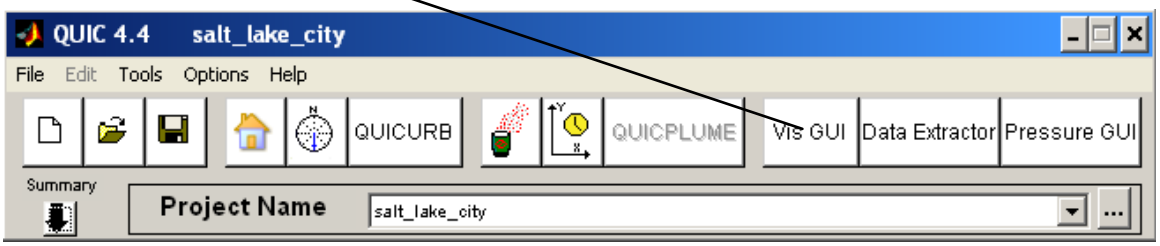
# VIS GUI

# QUIC Visualization GUI

- The Vis GUI first becomes activated once QUIC-URB has been run.

- Press

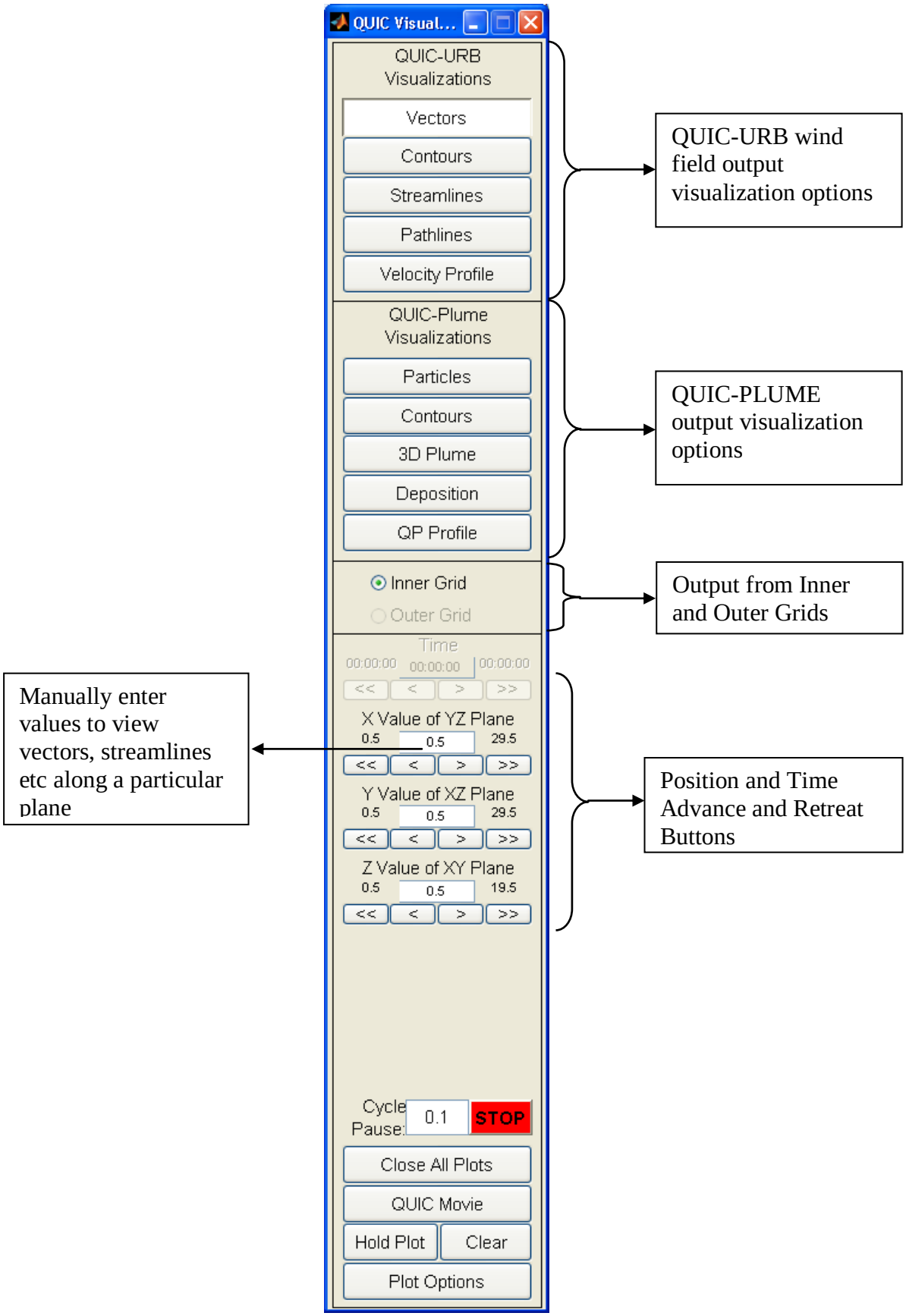
to open up the QUIC Visualization GUI as shown below.




- The Visualization GUI opens and it defaults to the wind vector plot selection shown below.

- The QUIC-URB wind field output can be visualized as wind vectors, contours, streamlines, pathlines, and velocity profiles.

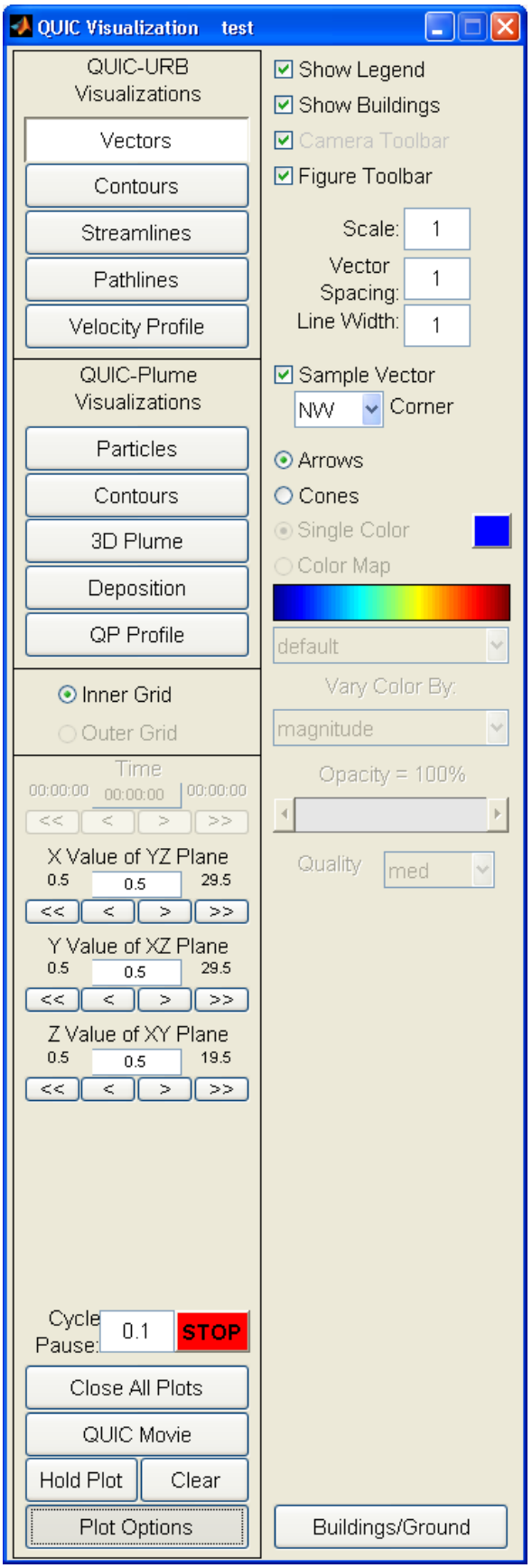
- The QUIC-PLUME model output can be visualized as particle positions, concentration, dosage, or LCT contours, isosurfaces, surface deposition contours, or concentration profiles.


QUIC Visual...
QUIC-URB Visualizations
Vectors
Contours
Streamlines
Pathlines
Velocity Profile
QUIC-Plume Visualizations
Particles
Contours
3D Plume
Deposition
QP Profile
Inner Grid
Outer Grid
Time
00:00:00
00:00:00
00:00:00
<< < > >>
X Value of YZ Plane
0.5
0.5
29.5
Y Value of XZ Plane
0.5
0.5
29.5
Z Value of XY Plane
0.5
0.5
19.5
Cycle Pause:
0.1
STOP
Close All Plots
QUIC Movie
Hold Plot
Clear
Plot Options
QUIC-URB wind field output visualization options
QUIC-PLUME output visualization options
Output from Inner and Outer Grids
Position and Time Advance and Retreat Buttons
Manually enter values to view vectors, streamlines etc along a particular plane

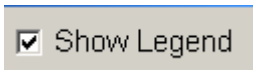
VIS-GUI provides the user a host of options to better view the results. For enhanced effects, press the 'Plot Options' button

at the bottom of the VIS GUI window

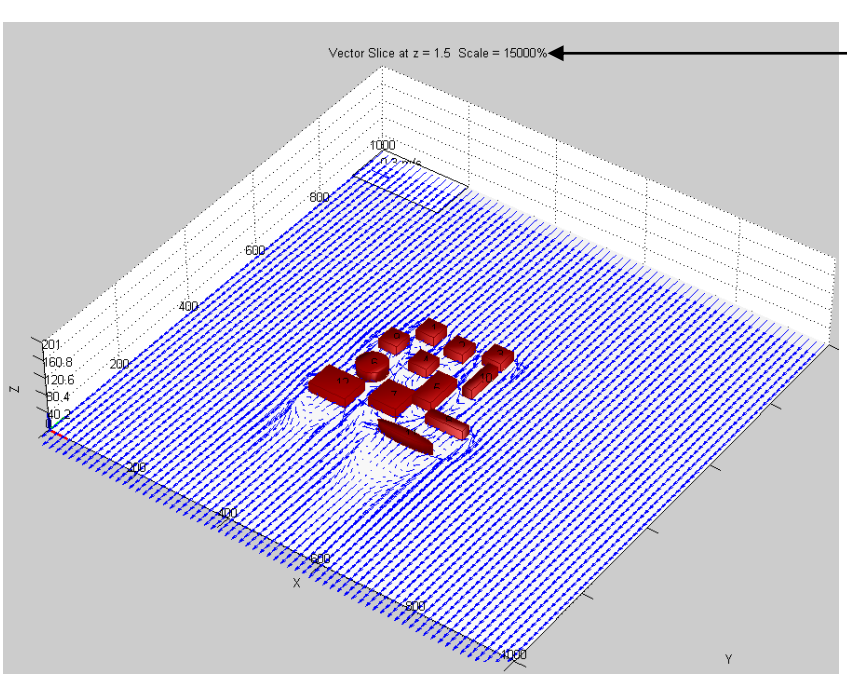
This launches a window with the various plot options as shown to the right. The plot options visible will change depending on which plot type is selected. The options shown are for vector plots. Some of the controls shown are common to several different plot types. This section will start with a description of the general plot controls and will later focus on the plot-type specific controls.

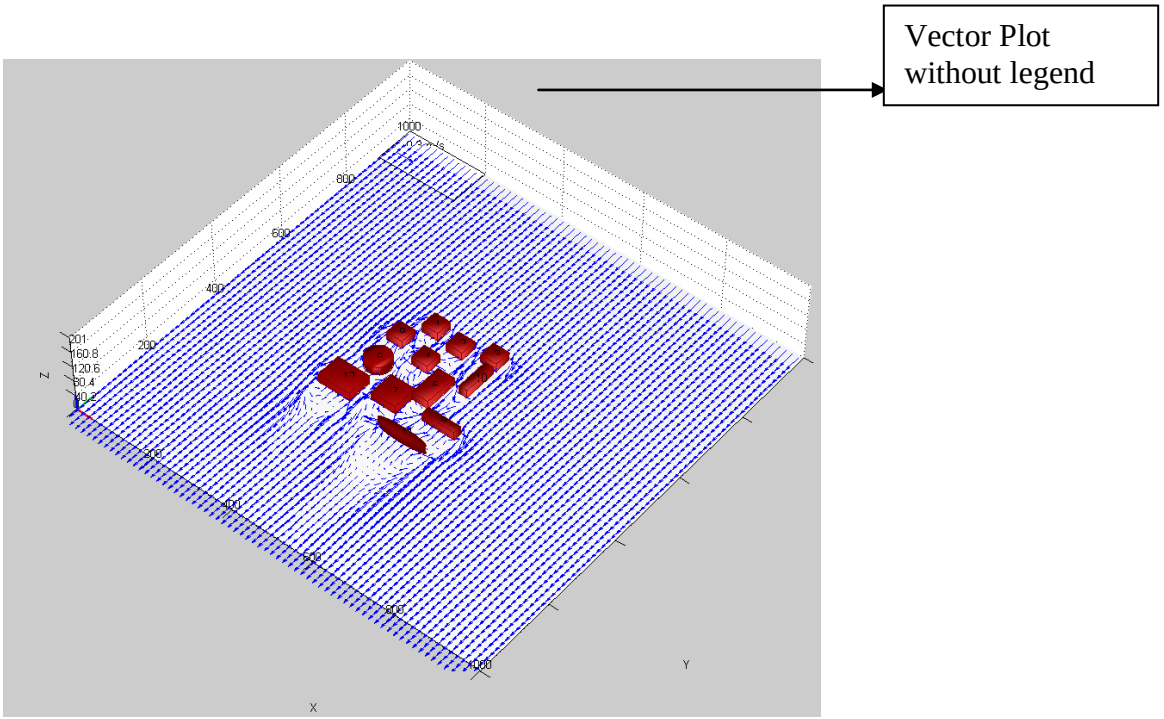


# Show Legend

Selecting the 'Show Legend' 

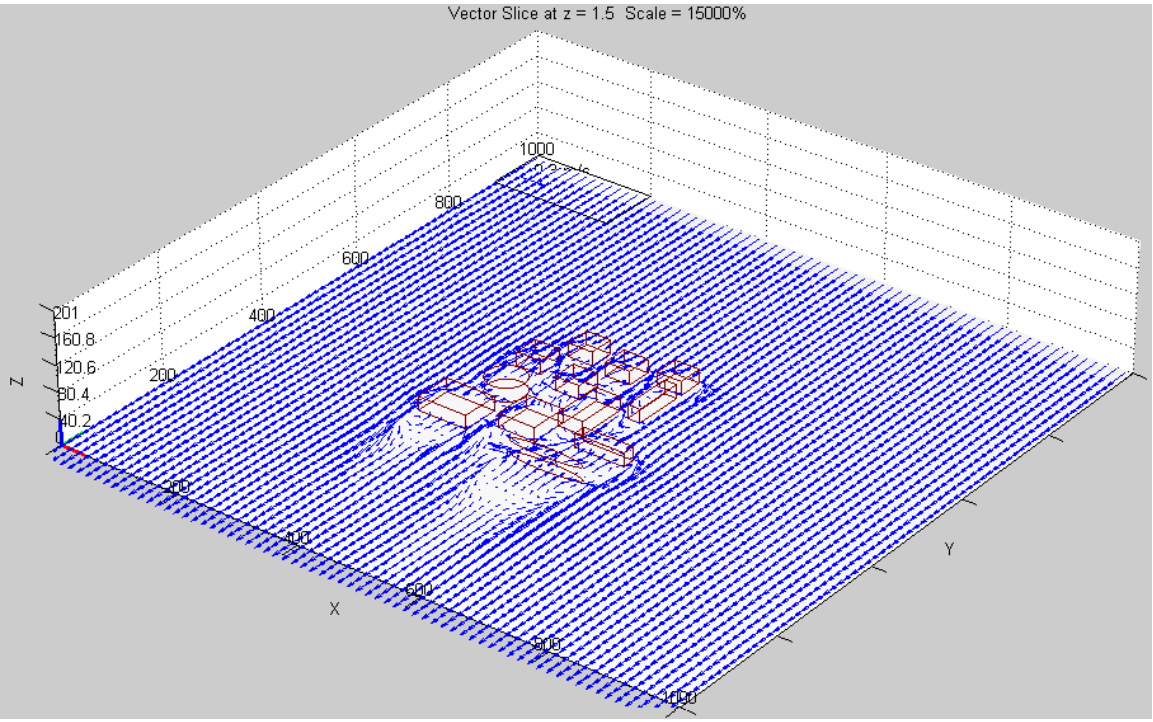
option places a legend on the vector plot. This also controls if a color bar is shown on the right hand side of the figure.



# Show Buildings

Selecting the 'Show Buildings' ☑ Show Buildings option displays the buildings in the vector plot. Deselecting it places wire frames instead of buildings. This is shown below

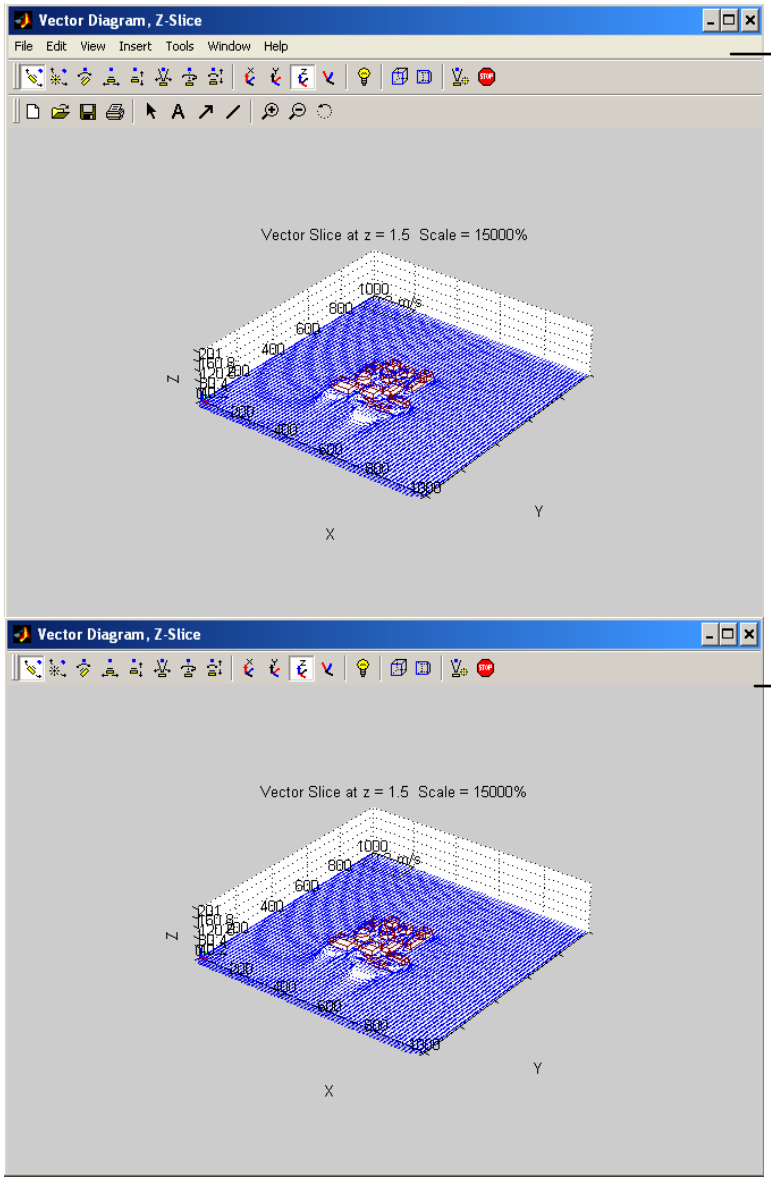



The figure above shows wire frames instead of buildings. This occurs when the 'Show Buildings' option is deselected.

# Figure Toolbar

Selecting the 'Figure Toolbar'

option displays figure toolbar in the vector plot MATLAB window. Deselecting it does not place it in the MATLAB window. This is shown below.



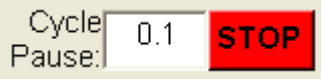

Selecting the 'Figure Toolbar' option

Deselecting the 'Figure Toolbar' option

## Time and Position Controls

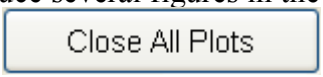
Time and position can be selected by either entering the desired value in the time and position edit boxes or by using the advance and retreat buttons.

The single arrow buttons advance/retreat by a single increment. The double arrow buttons cycle through all possible values from the current time/location to the maximum or minimum value, depending on directionality of the button. The length of time (in seconds) that the GUI pauses in between cycles is set in the 'Cycle Pause' edit box.




The 'Stop' button causes the plots to stop cycling.

## Close All Plots

Since each different plot type is generally placed within its own figure window, it is easy to produce several figures in the process of visualizing QUIC data. The 'Close All Plots' button 

will close all of the figure windows produced by the Visualization GUI. This function is also executed when the Visualization GUI is closed.

## Saving Animations

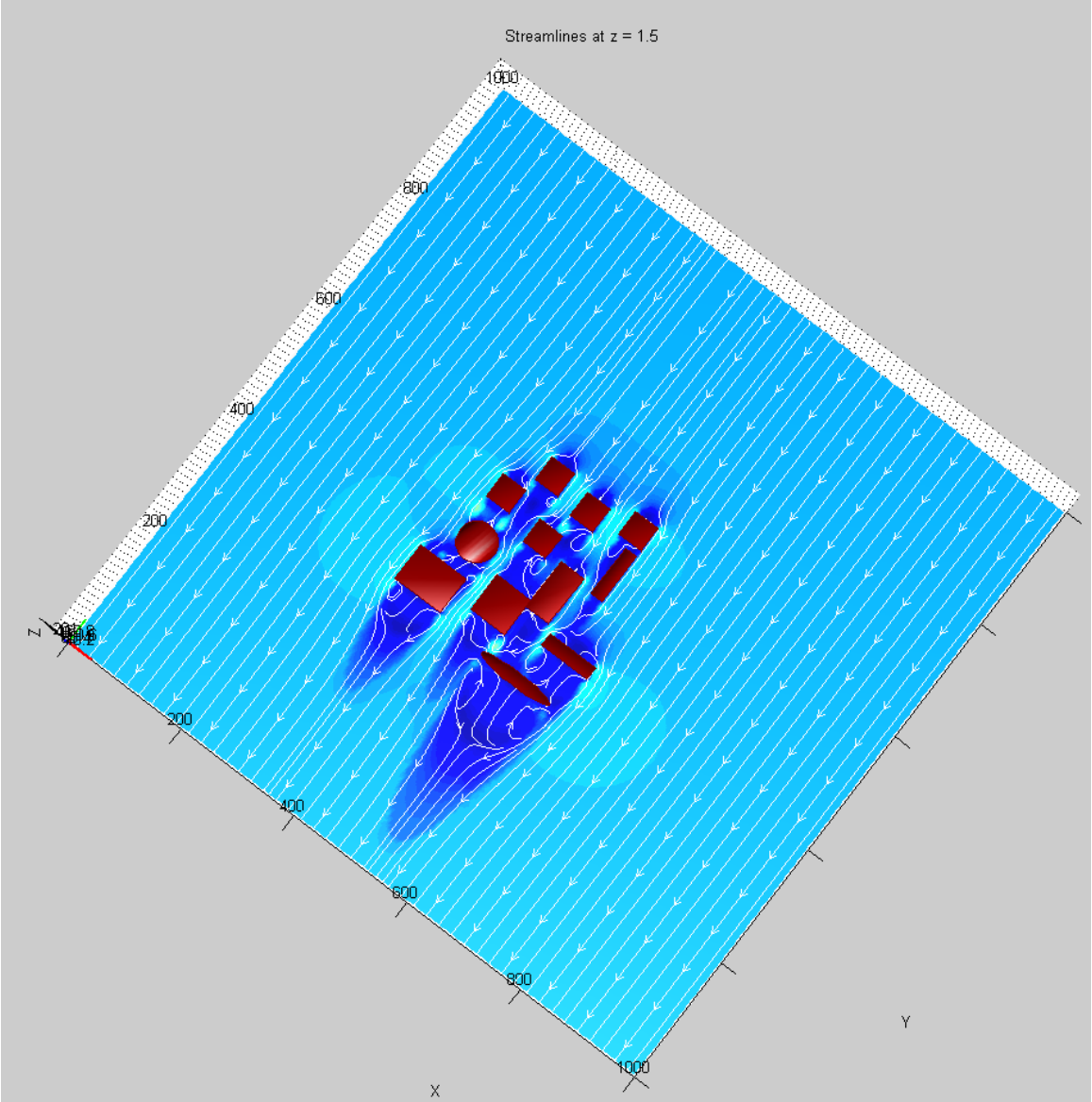
The 'QUIC Movie' button

launches the QUIC Movie GUI which produces animations from the plots produced by the Visualization GUI. The details regarding the procedures of movie creation will be discussed later in the QUIC Movie GUI section.

# Hold Plot

Multiple plots can be placed on a single figure using the 'Hold Plot' button

. After producing a plot pressing the 'Hold Plot' button will hold the current plot and cause future plots to plot to the held figure. Multiple plots can be held on a single figure. Below is an example of the combination of velocity magnitude and streamline plot produced using the hold plot button.



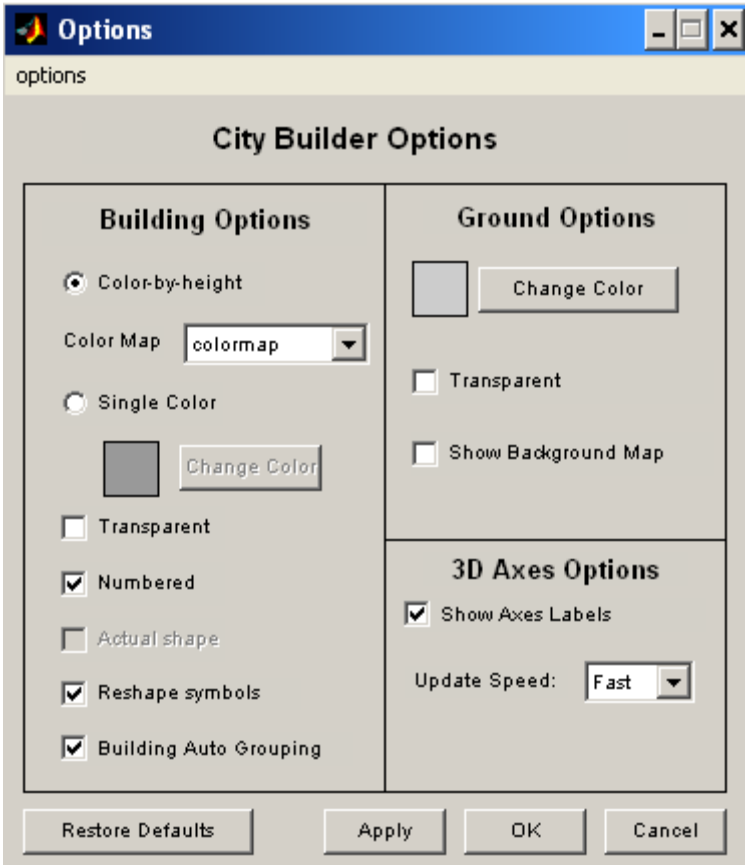

Pressing the 'Clear' button

will delete the held figure and will cause the GUI to plot different plot types in different figures.

# Buildings / Ground

The Buildings / Ground button

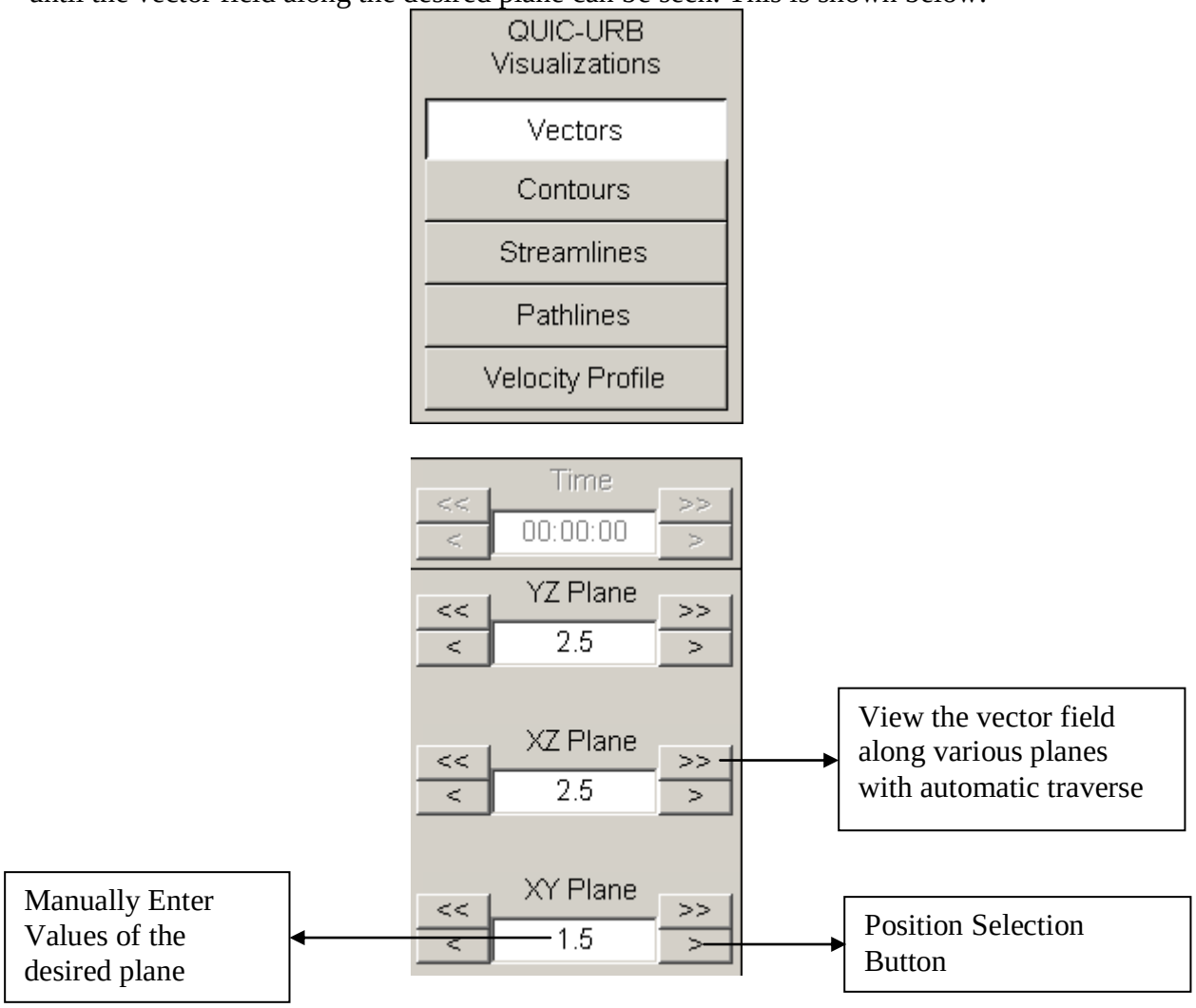
found at the bottom of the VIS GUI window is the same as the one in the City Builder Window. It enables the user change the color of the buildings and the ground; add transparency to both these entities and a host of other options. For these options, please refer the City Builder section.

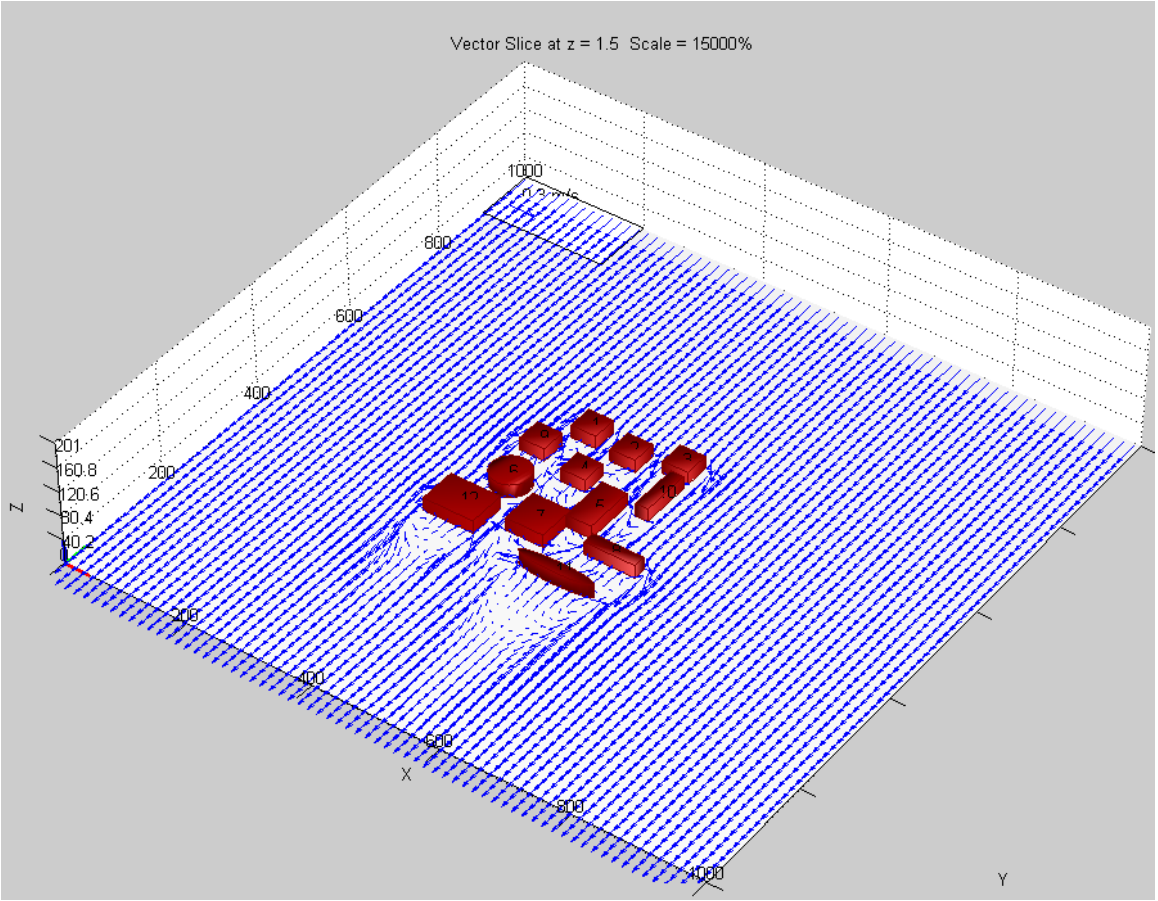


## QUIC-URB VISUALIZATIONS

### Vectors

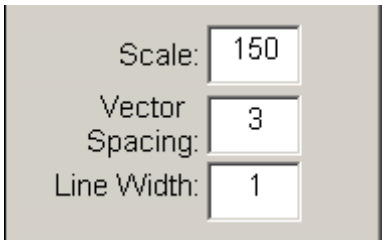
To view the vector field along a particular plane, hit the 'Vectors' button and enter the value of the x-y, x-z, or y-z plane or use the 'Position or Time Selection' buttons > until the vector field along the desired plane can be seen. This is shown below.






Several plot options apply

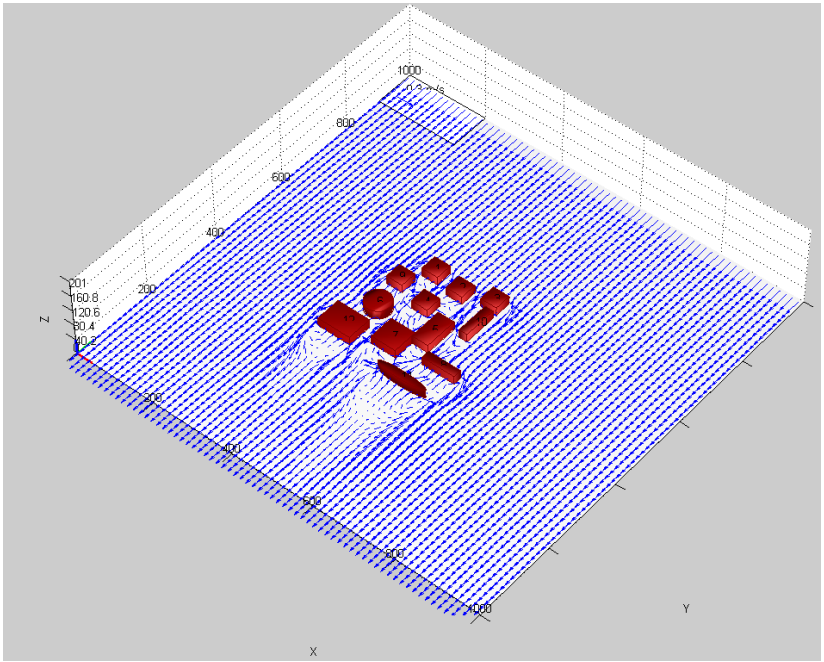
## *Scale, Vector Spacing and Line Width*



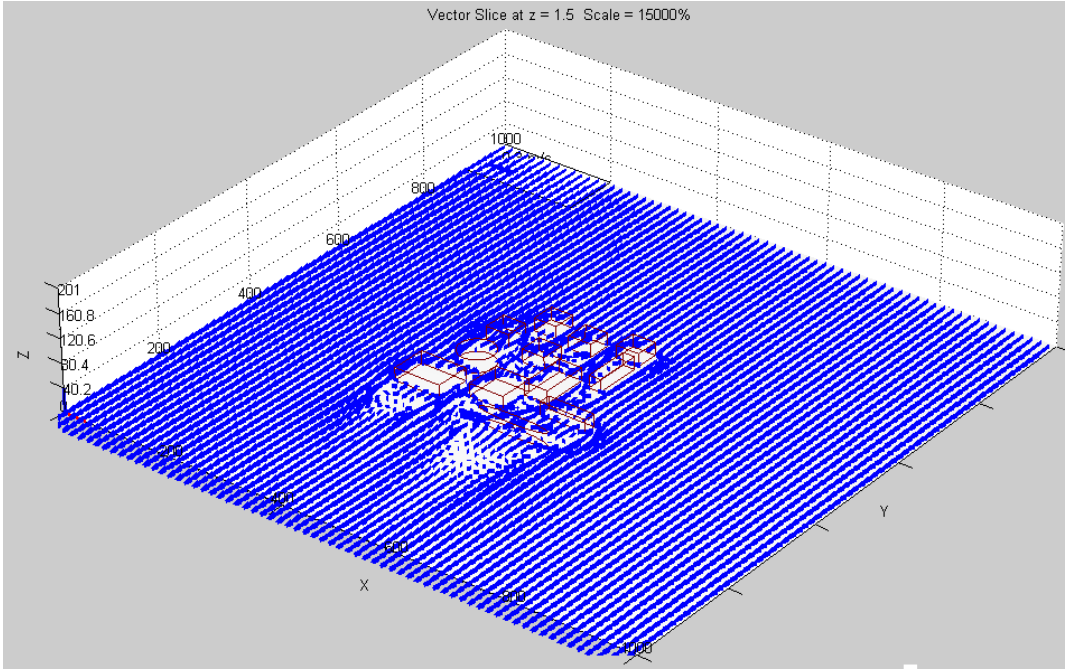

Vector spacing determines the number of grid cells between vector arrows, i.e., a vector spacing of 2 means that 1 arrow is placed every 2 grid cells.

The scale value is multiplied by the vector magnitude to magnify smaller vectors, note that 0 = 1 = no scaling

Line width increases or decreases the line width of the vectors in the vector plot. The figures below show the difference between vector plots with Line Width = 1 versus Line Width = 3.



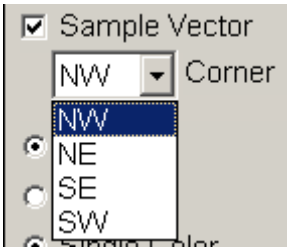

Vector Plot with Line Width = 1

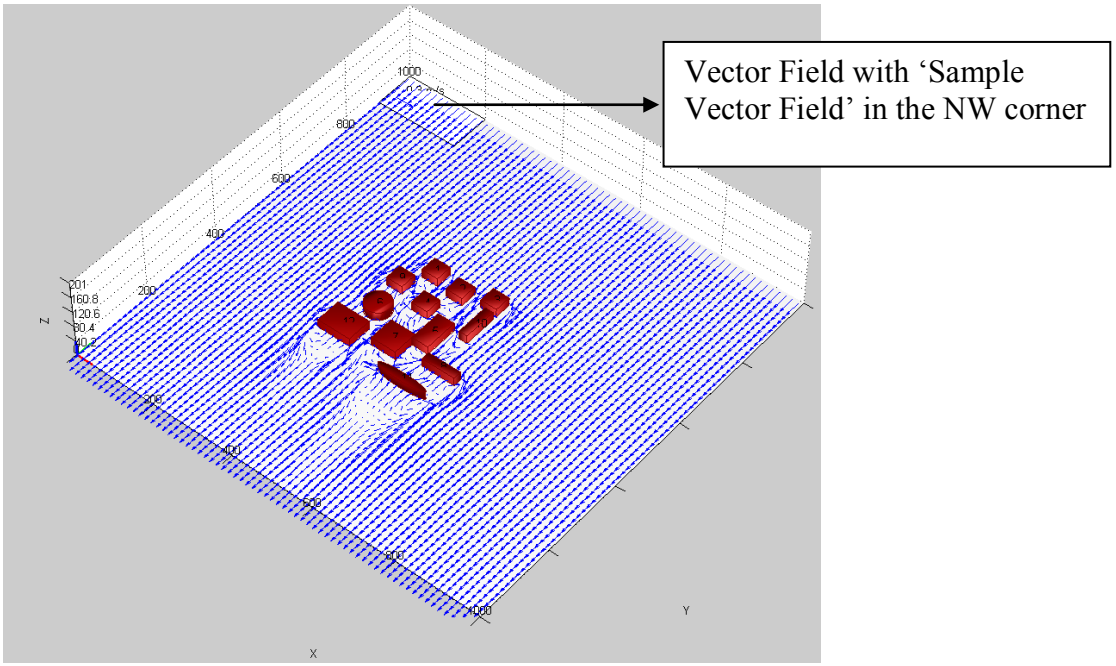



Vector Plot with Line Width = 3

## *Sample Vector*

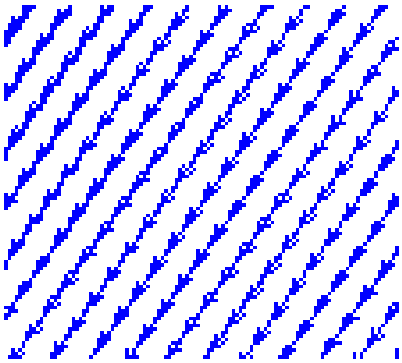
Selecting the 'Sample Vector' option places a sample vector field on any of the four corners of the vector plot. Deselecting the 'Sample Vector' option does not place a sample vector field. The user can choose to place the sample vector field on any of the four corners, i.e. North-East, North-West, South-East or South-West. This is illustrated in the figures below.





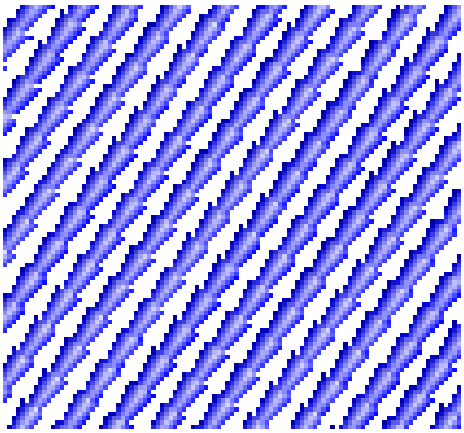

## *Arrows*

Selecting the 'Arrows' option places arrows as vector heads. This is shown below.



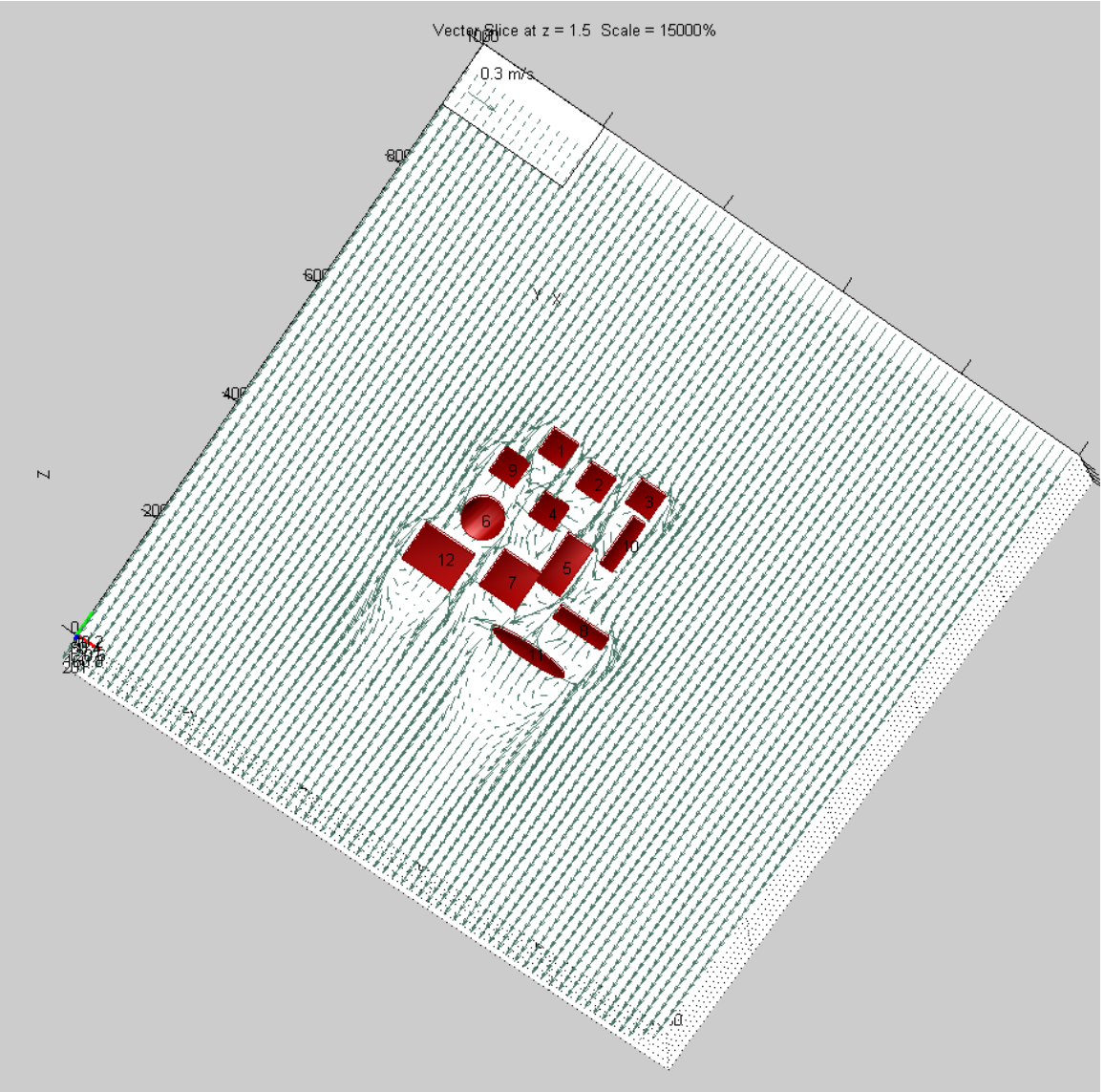
## *Cones*

Selecting the 'Arrows' option places arrows as vector heads. This is shown below.



## *Single Color*

Selecting the 'Single Color' option plots the vector field without any contour levels. That is to say that the color does not represent the magnitude of the velocity. The user has the option of choosing the color of the vector field. To do so, click the color button. This launches the 'Set Color' window. Using this, choose the color of the vectors in the vector plot. This is shown below.

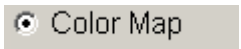


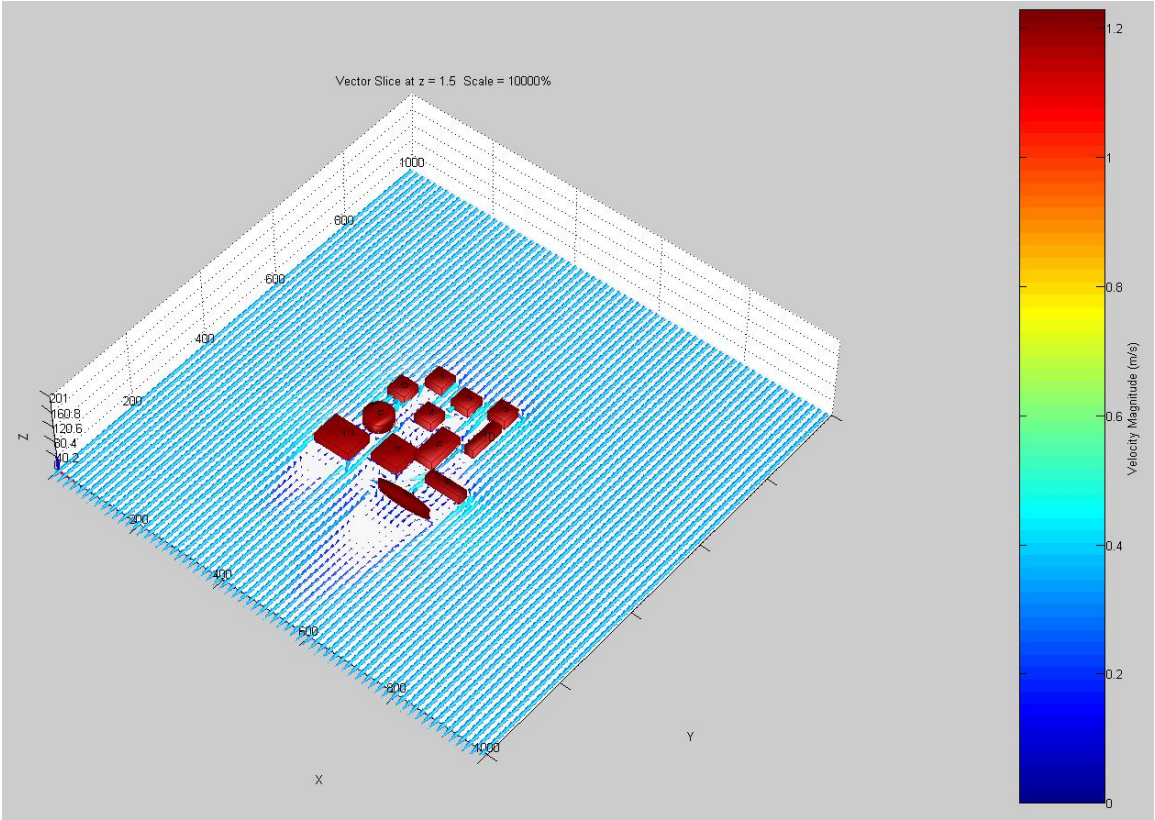

The figure above shows the vector plot with the green color vectors

Note: The ‘Single Color’ option can be used both the ‘Arrows’ and ‘Cones’ options

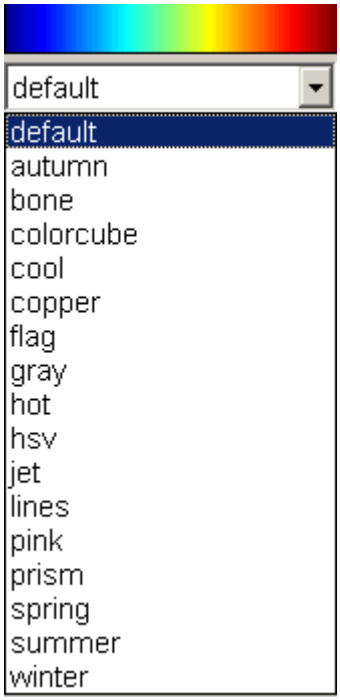
## *Color Map*

The ‘Color Map’ option at present works only with the ‘Cones’ option. Selecting the ‘Color Map’ option plots the vector field with contour levels. That is to say that the color of the vector field is representative of the magnitude of velocity. This is shown in the figure below.



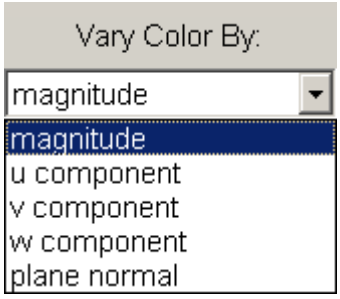



The user has the option of picking colors for the color map. This can be done through the pull down menu as shown below.



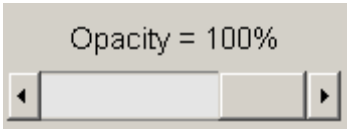
## *Coloring Cones by Various Quantities*

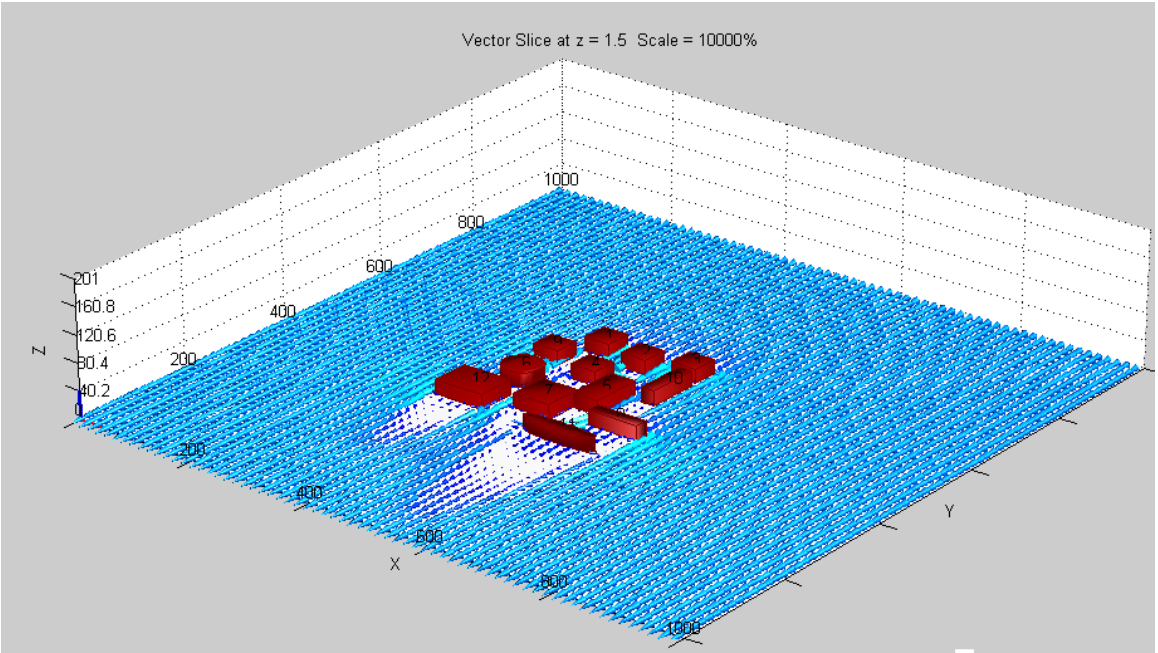
The user has the option of viewing the vector field based in the magnitude of velocity or based on the u, v, or w components of velocity. This can done using the pull down menu as shown below.



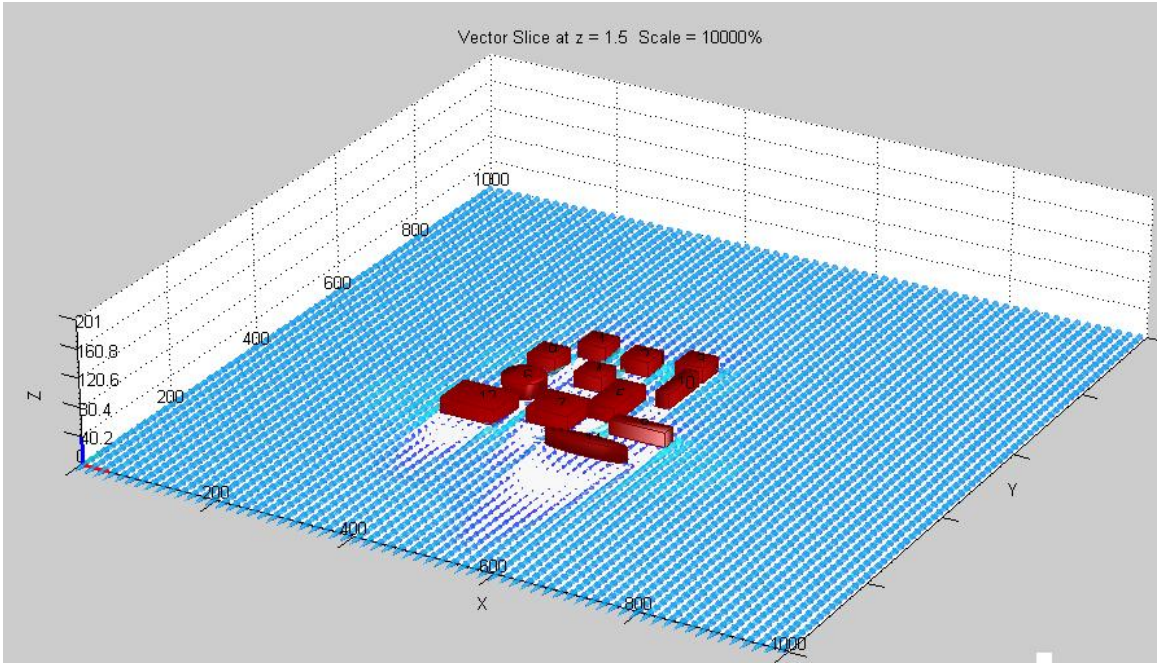

## *Opacity*

The opacity option can be used to add transparency to the vector field. By this it follows that 0% opaque would make the vector field invisible. Opacity can be changed with the slide bar as shown below.

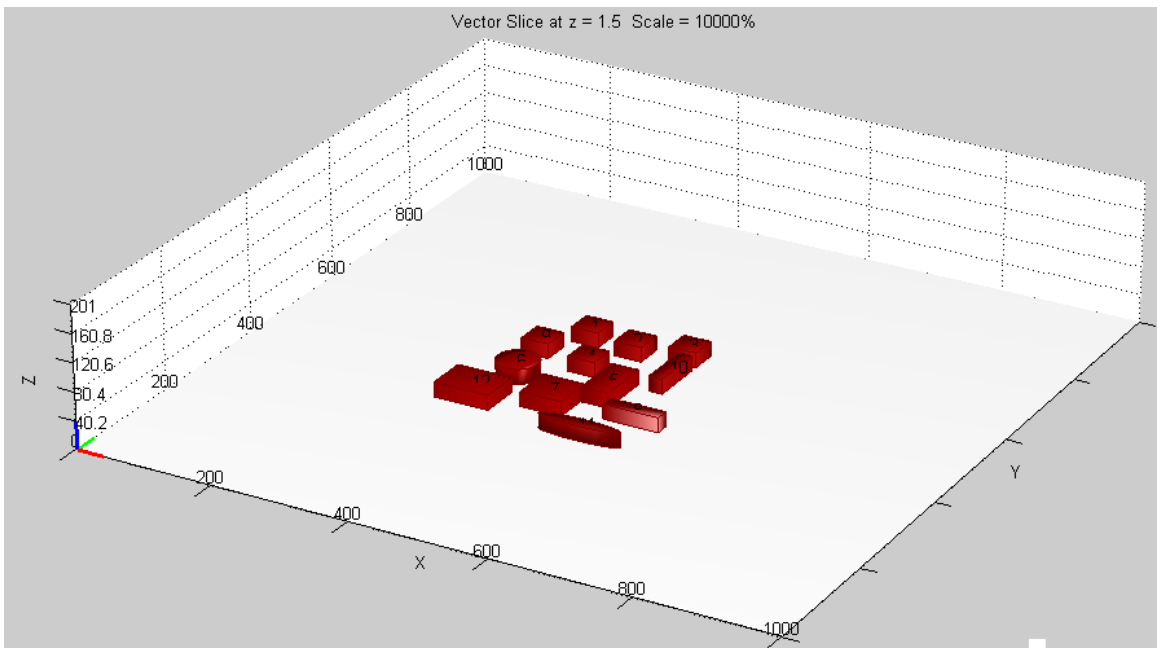



**Opacity = 100%**



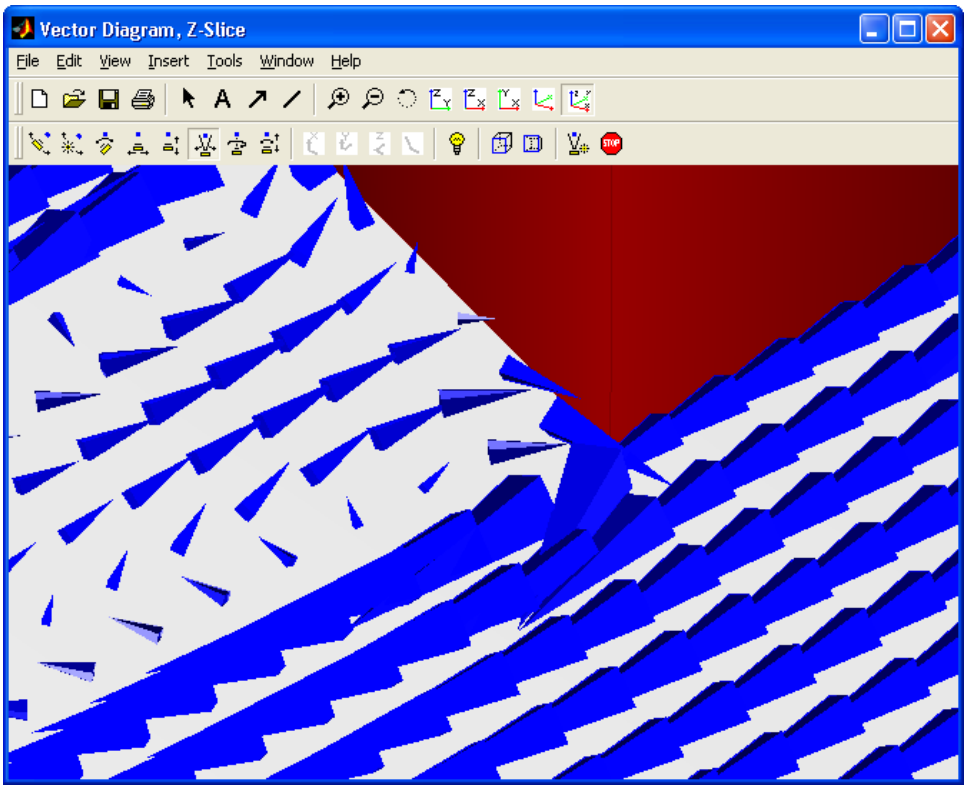

**Opacity = 50%**



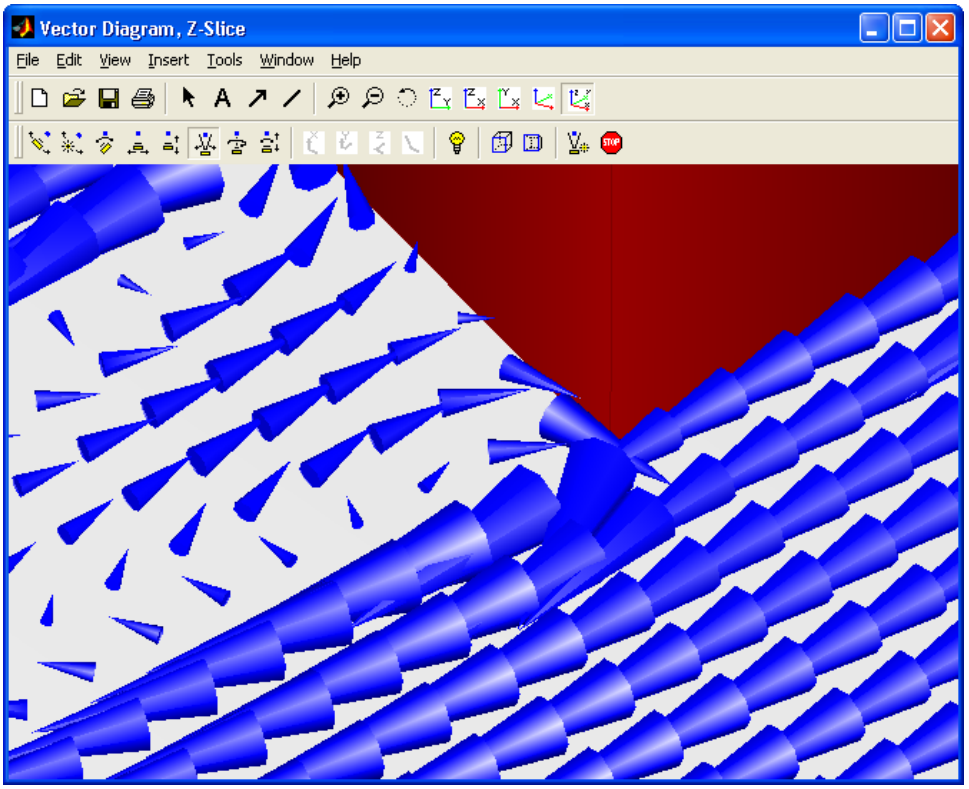

**Opacity = 0%**

## *Quality*

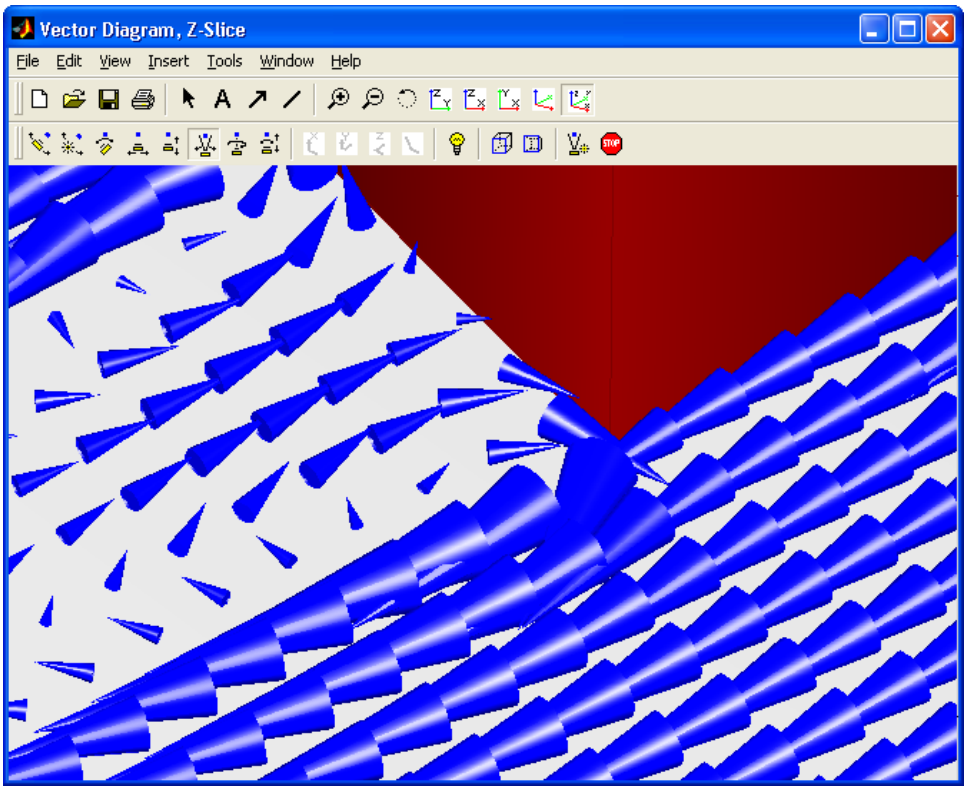
Choosing 'Quality' determines the resolution of the graphics used to display the plot. The 'Quality' pull down menu gives the user three options; of choosing between High, Medium and Low. Though choosing 'high' gives better graphical resolution, it is more time intensive. Choosing 'Low' gives the user the plot faster but with lesser graphical resolution. This is due to the computation being done on the fly. The differences in the plots when each of these options is chosen are shown below.




Choosing 'Quality' as 'Low'



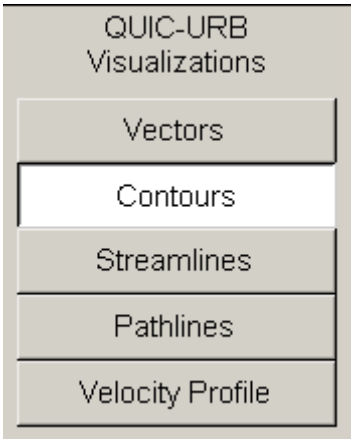

Choosing ‘Quality’ as ‘Medium’



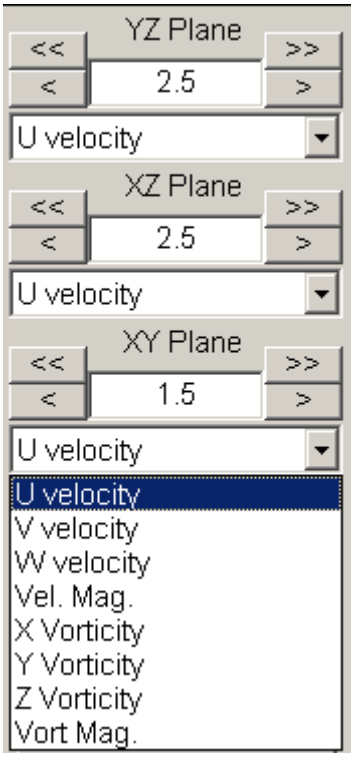

Choosing ‘Quality’ as ‘High’

# Contours

The resultant vector field from QUIC URB can be visualized as contour plots by pressing the contour button under QUIC-URB Visualizations

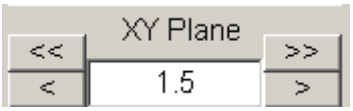


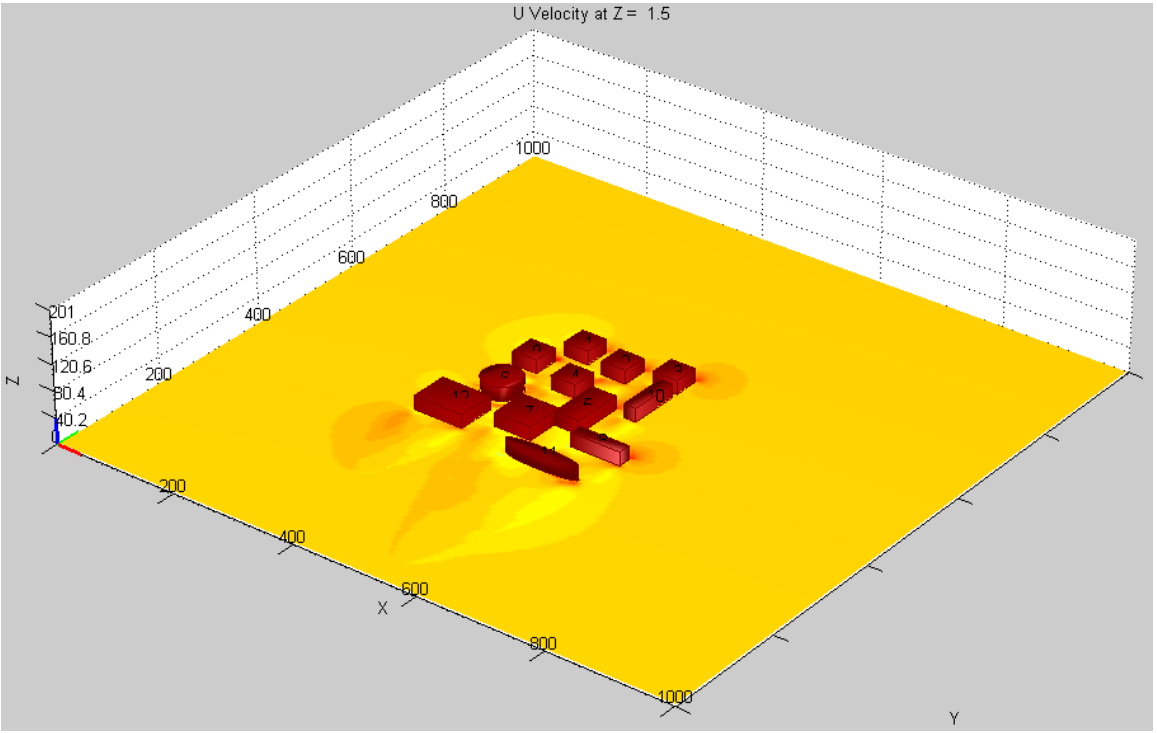

VIS-GUI provides the user the option of viewing the contours of all three components of velocities and also the contours of quantities derived from the mean velocity field (vorticity etc). This is depicted in the figure below.



Note: While vorticity is one of the default plot options it is not calculated by default. This is because this can be computationally expensive, particularly for large domains. Instead the vorticity is computed the first time that the user plots the vorticity field and is saved in 'vorticity.mat' for later use. Thus it will likely take much longer to plot vorticity the first time than it will in subsequent attempts. This plot option can also be used to plot the turbulence fields (velocity sigmas, TKE, velocity covariances, and U*) if the 'QP_turbfield' output file option was selected before running QUIC-PLUME.

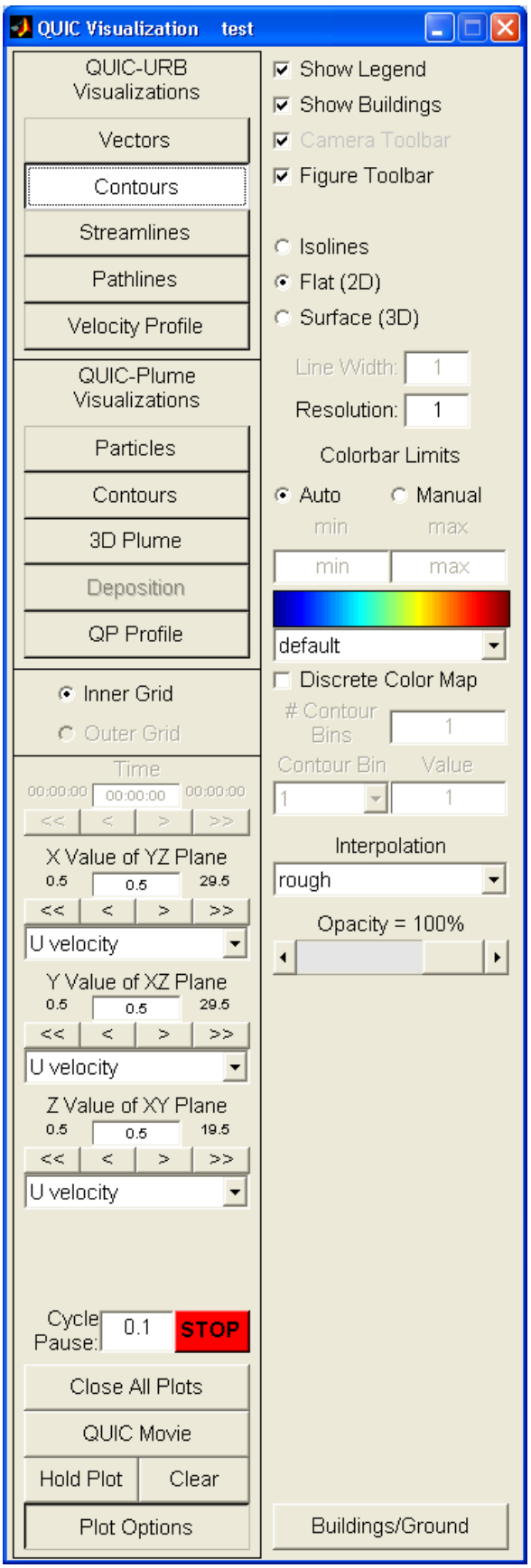
To view the contour plot along a particular plane, enter the value of the x-y, x-z, or y-z plane or use the 'Position Selection' button > until the contour along the desired plane can be seen. This is shown below.



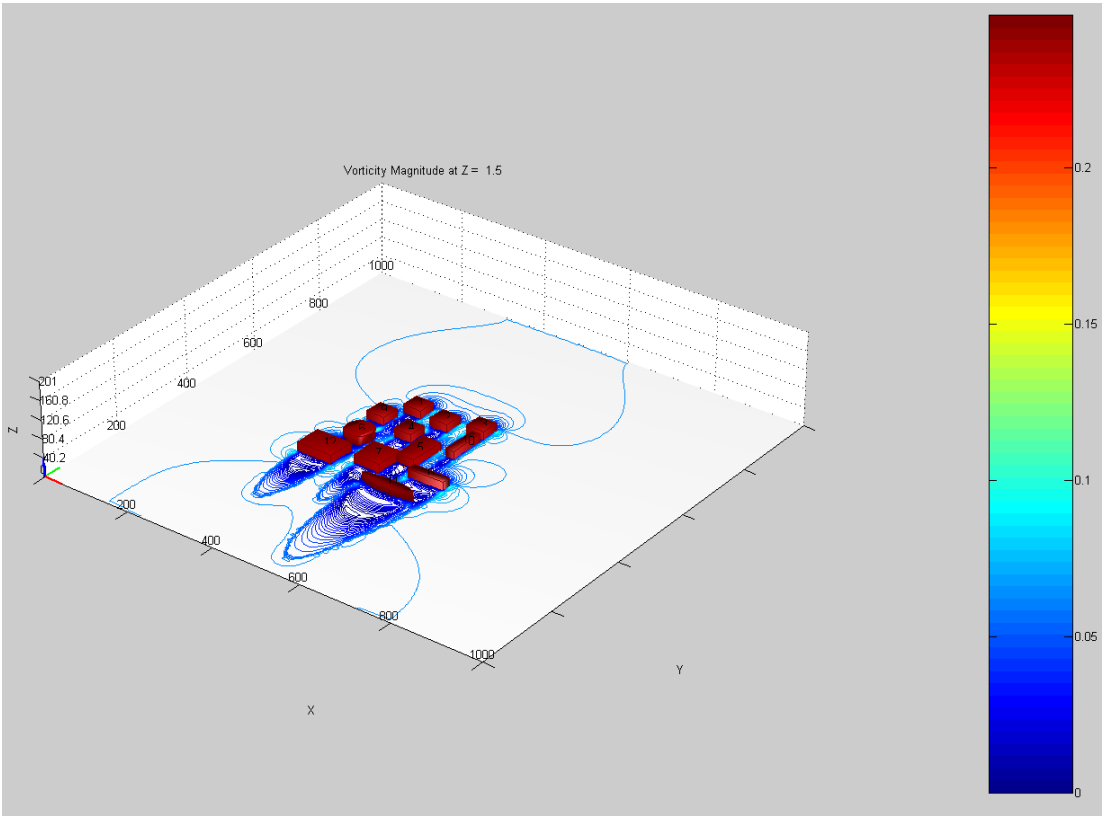
For enhanced effects, press the 'Plot Options' Plot Options button and this brings up a host of options for better visualizing contour plots. This is shown in the figure to the right.

The first three options, i.e. 'Show Legend', 'Show Buildings' and 'Figure toolbar' are the same as the ones described in the previous sections.



# *Isolines*

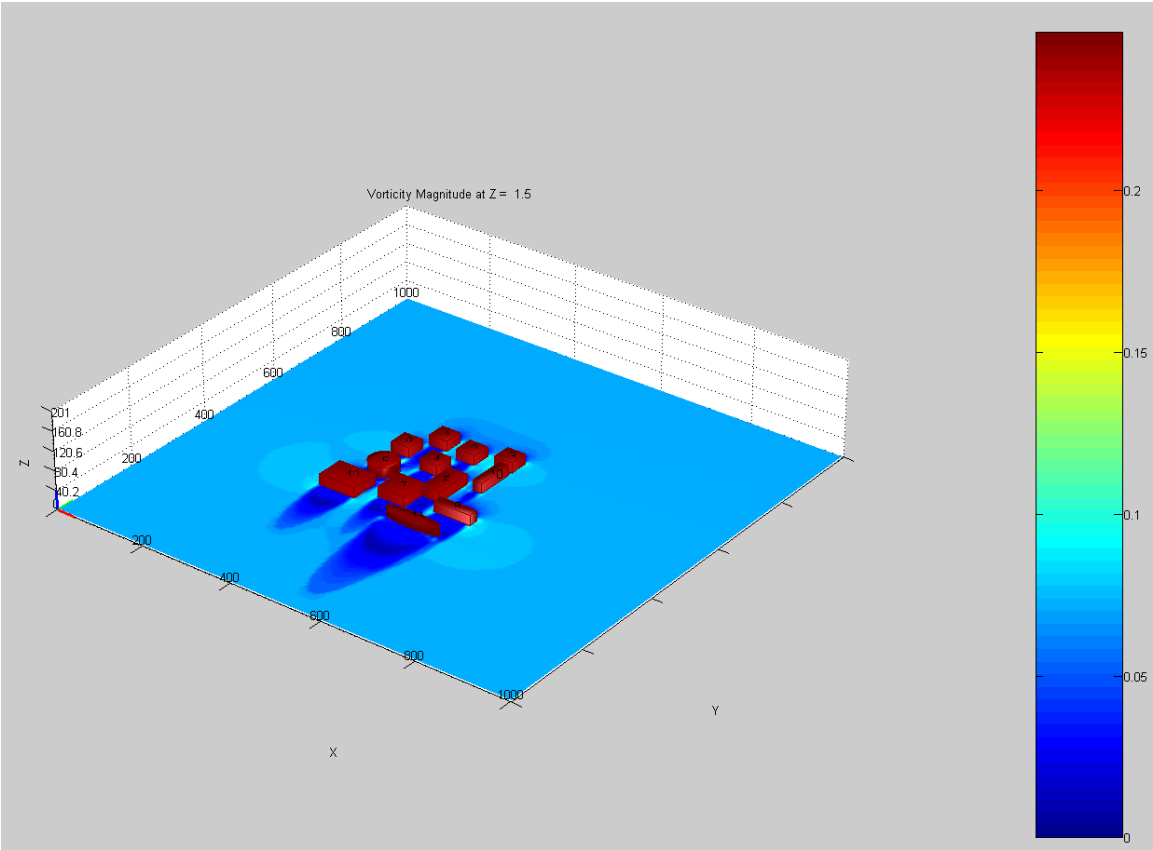
Selecting the 'Isolines' option draws isolines in the domain for the velocities or their derived quantities. This is shown below.




Isolines showing Vorticity Magnitude

## *Flat (2D)*

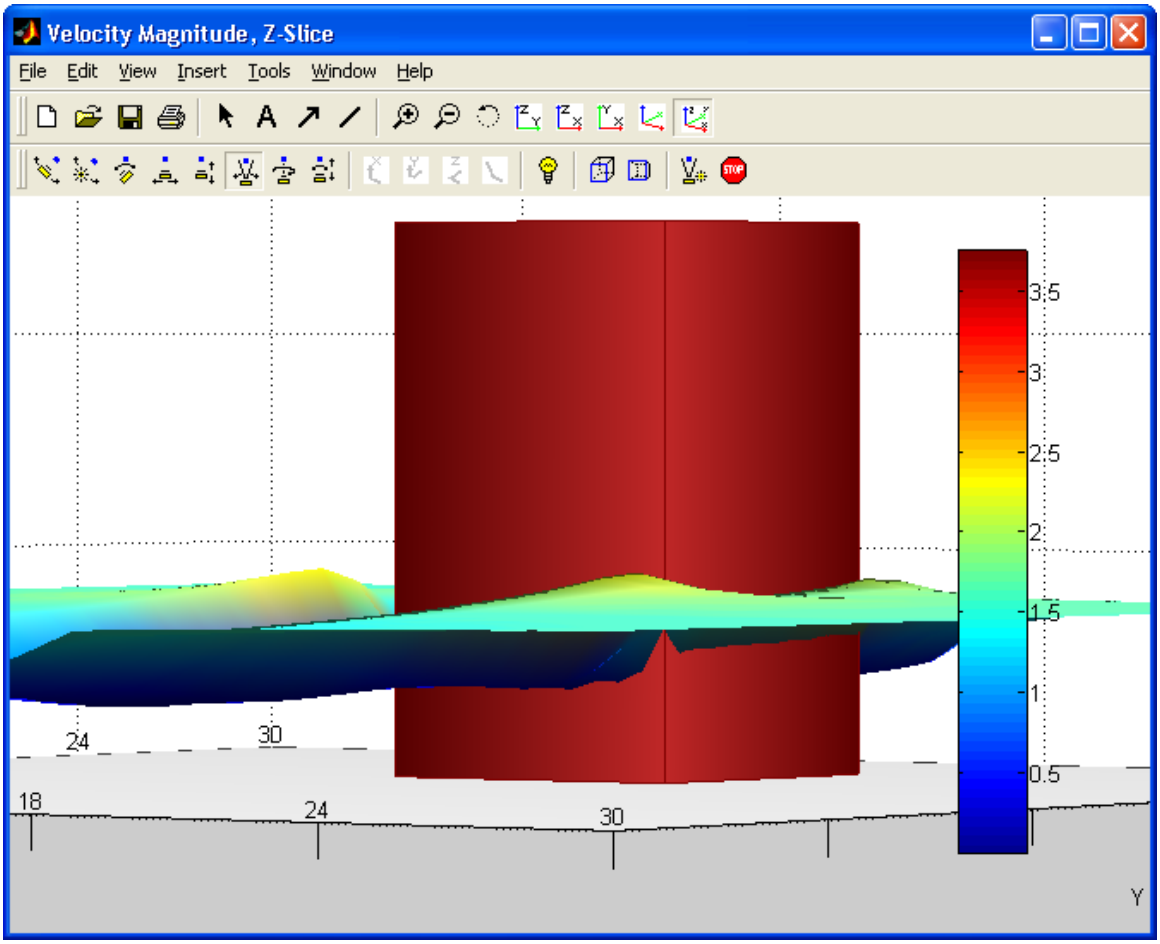
Selecting the Flat 2D option plots 2D solid contours of velocities or their derived quantities. This is shown below.




2D Contour Plot of Vorticity Magnitude

## *Surface (3D)*

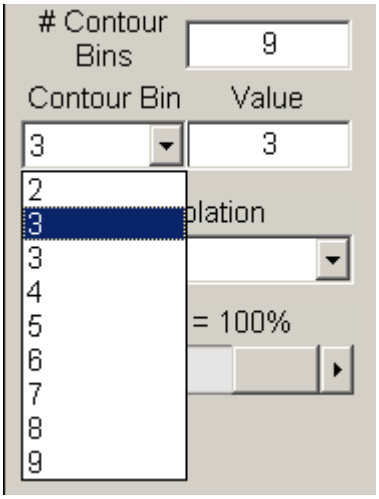
Selecting the Surface 3D option produces a 3D surface plot of the contours as is shown below.

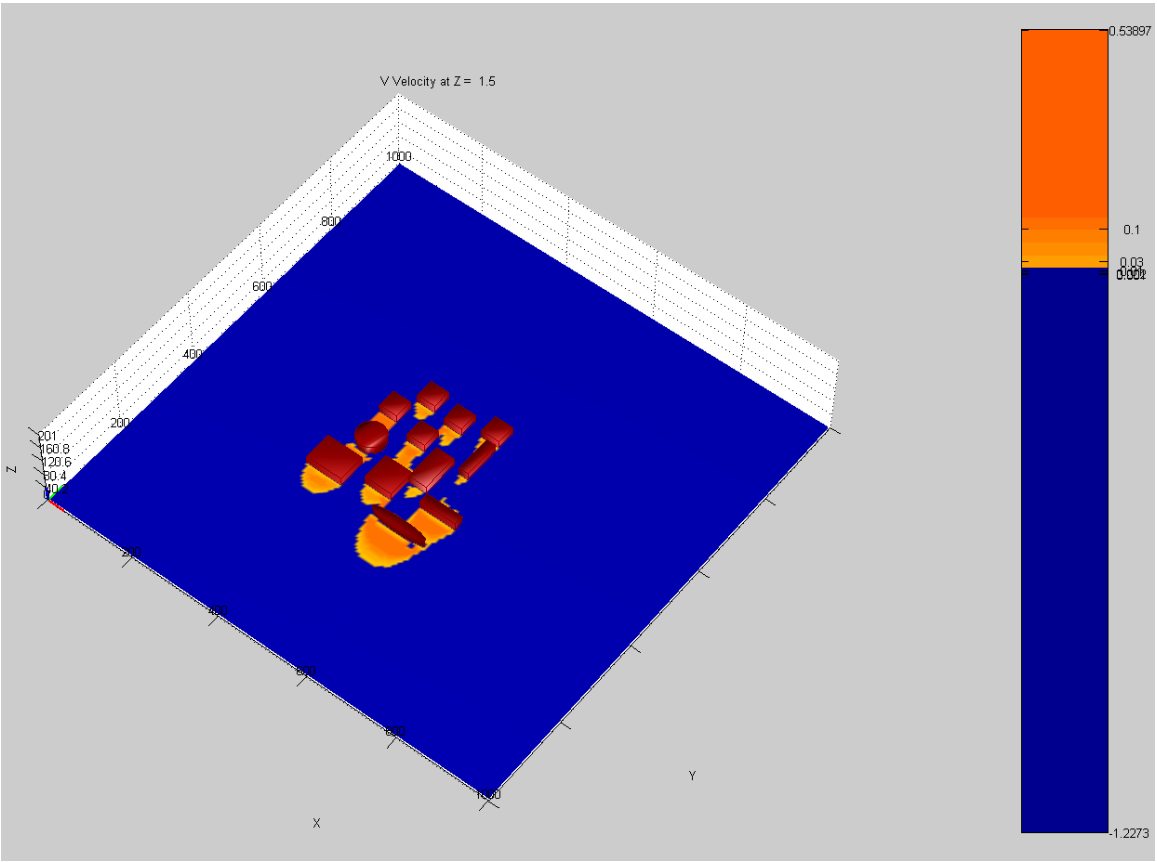



3D Contour Plot of Velocity Magnitude

## *Discrete Color Map*

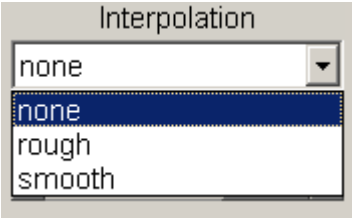
'Discrete Color Map' gives the user the option of specifying the contours levels. To do so, enter the number of contour bins that have particular value.

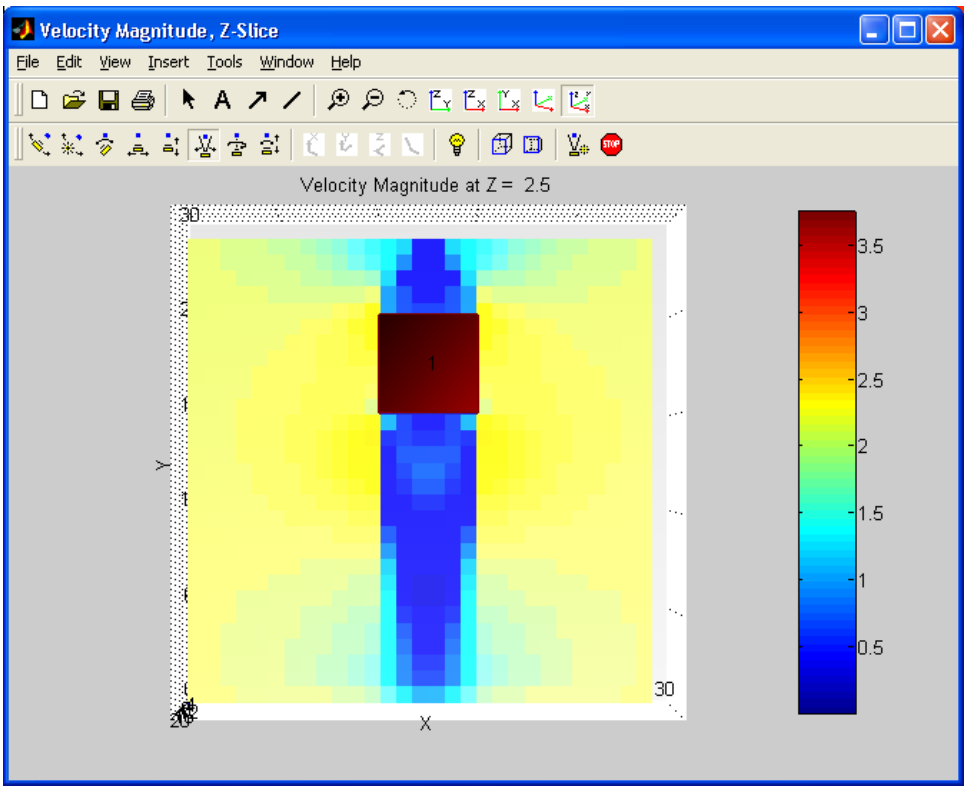


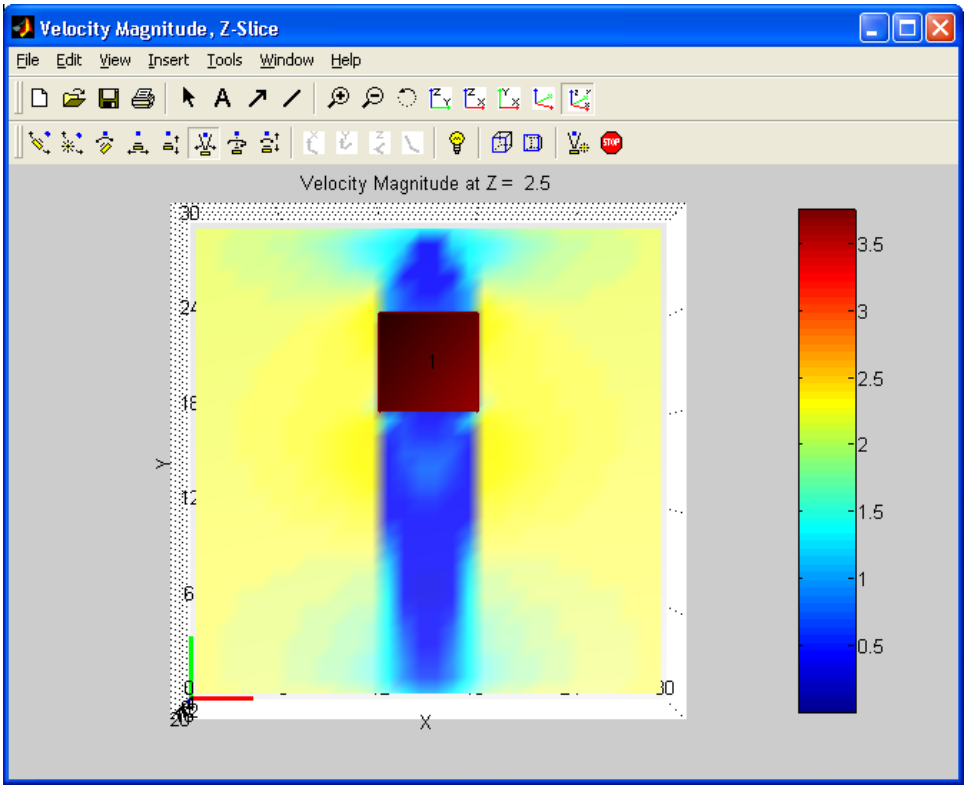
## *Interpolation*

The interpolation option interpolates between the various contour levels and produces smoother contour plots. There are there interpolation options; none, rough and smooth. As was the case with 'Quality - High', choosing 'Smooth' takes longer time for plotting but gives higher graphical resolution. This is illustrated in the plots below

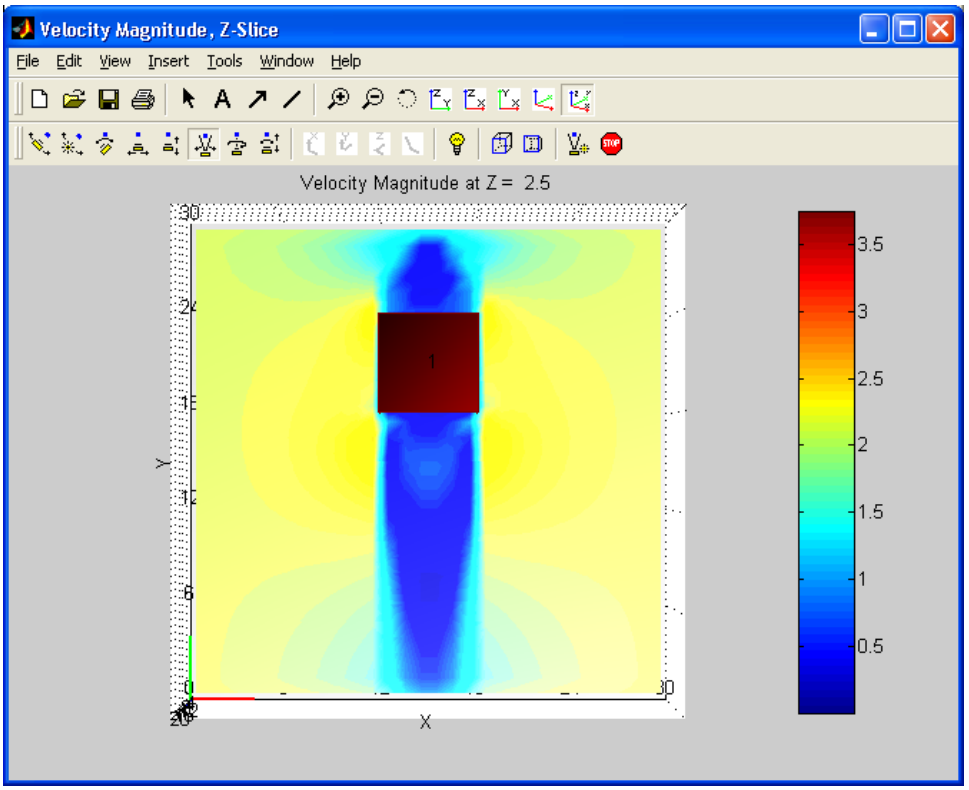



Interpolation - None



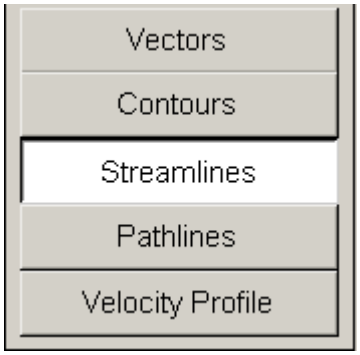

Interpolation – Rough

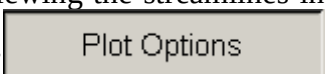



Interpolation – Smooth

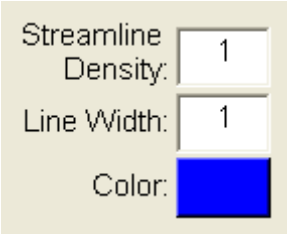
# Streamlines

The resultant vector field from QUIC URB can be visualized as streamlines by pressing the streamlines button under QUIC-URB Visualizations



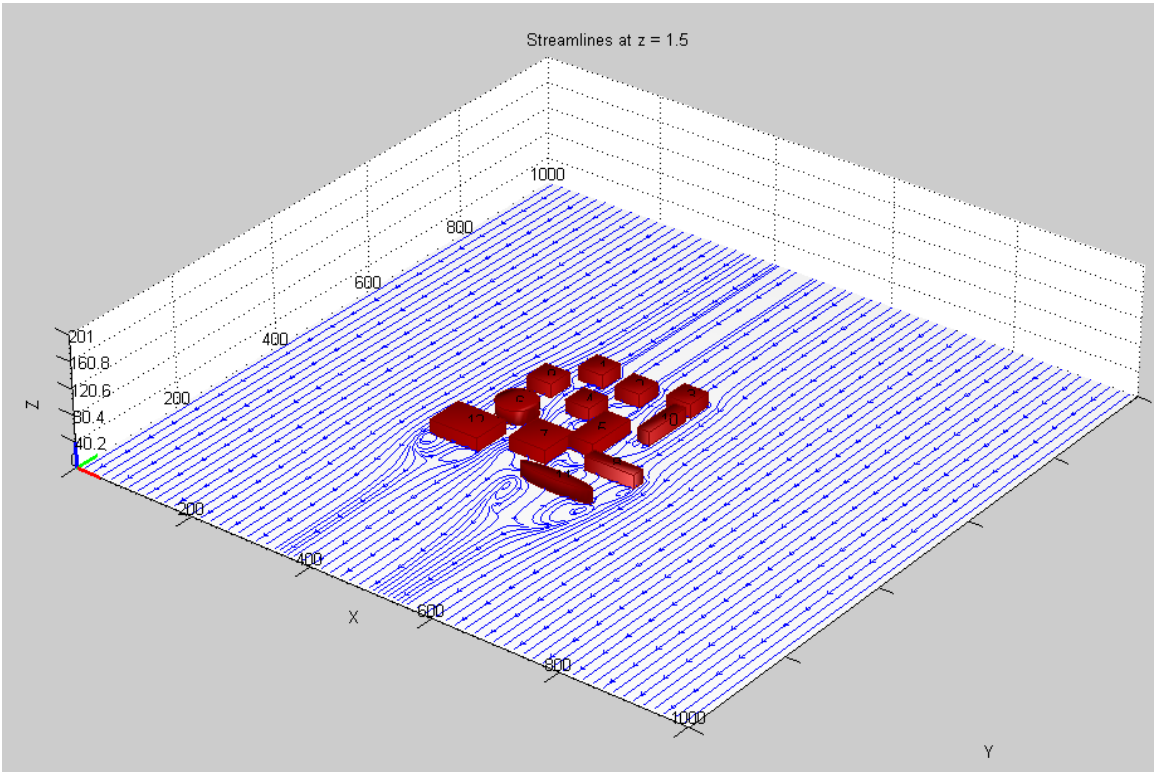

VIS-GUI provides the user the option of viewing the streamlines in various planes. For additional plot options, press 'Plot Options' 

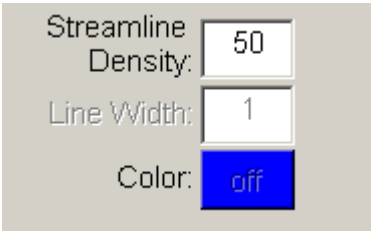
as was described in the previous sections. This launches a window with additional streamlines plot options. The various plot options for streamlines are shown below.

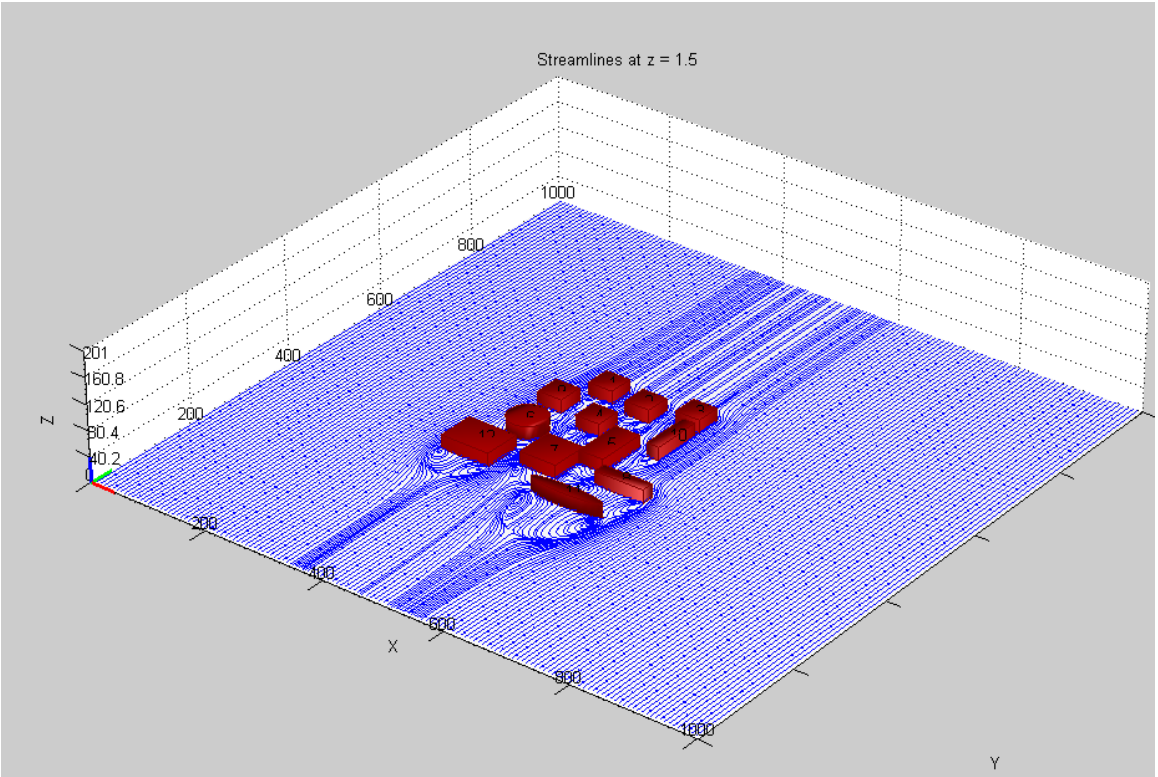



## *Streamline Density*

Streamline Density provides the user the option of increasing or decreasing the density of streamlines in the plots. This is shown below.

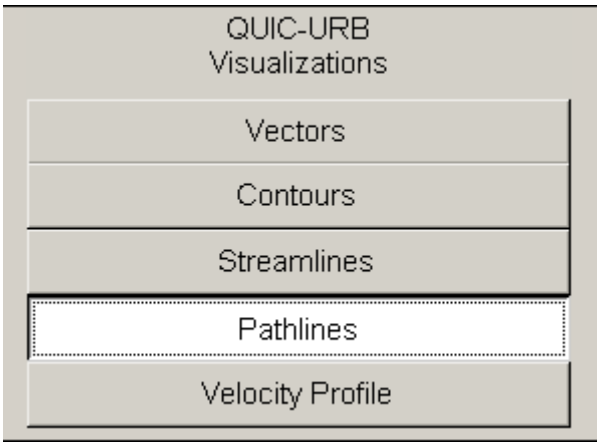

Streamlines at z = 1.5
1000
800
600
400
200
201
160.8
120.6
80.4
40.2
Z
200
400
600
800
1000
X
Y


Streamline Density:
50
Line Width:
1
Color:
off

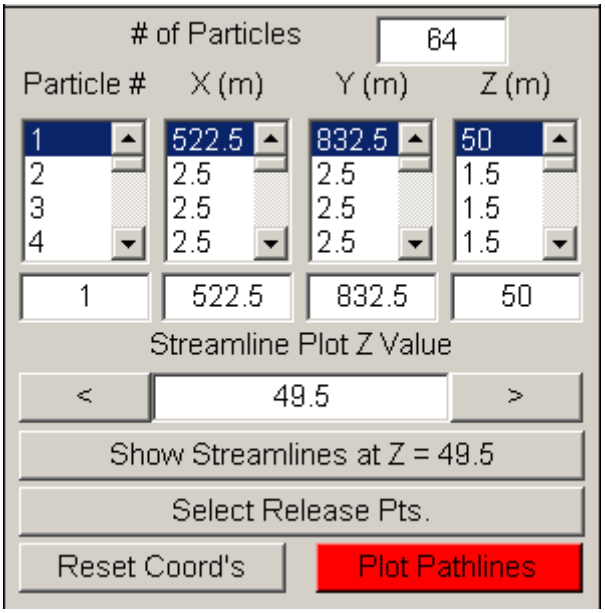



# Pathlines

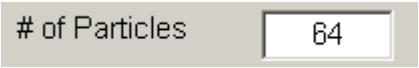
Pathlines of the wind field can be visualized by pressing the pathlines button under QUIC-URB Visualizations

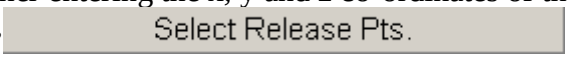



This bring up a window with pathline options




The maximum number of particles that can be released is 64. Enter the number of particles between 1 and 64 in the 'Number of Particles' window

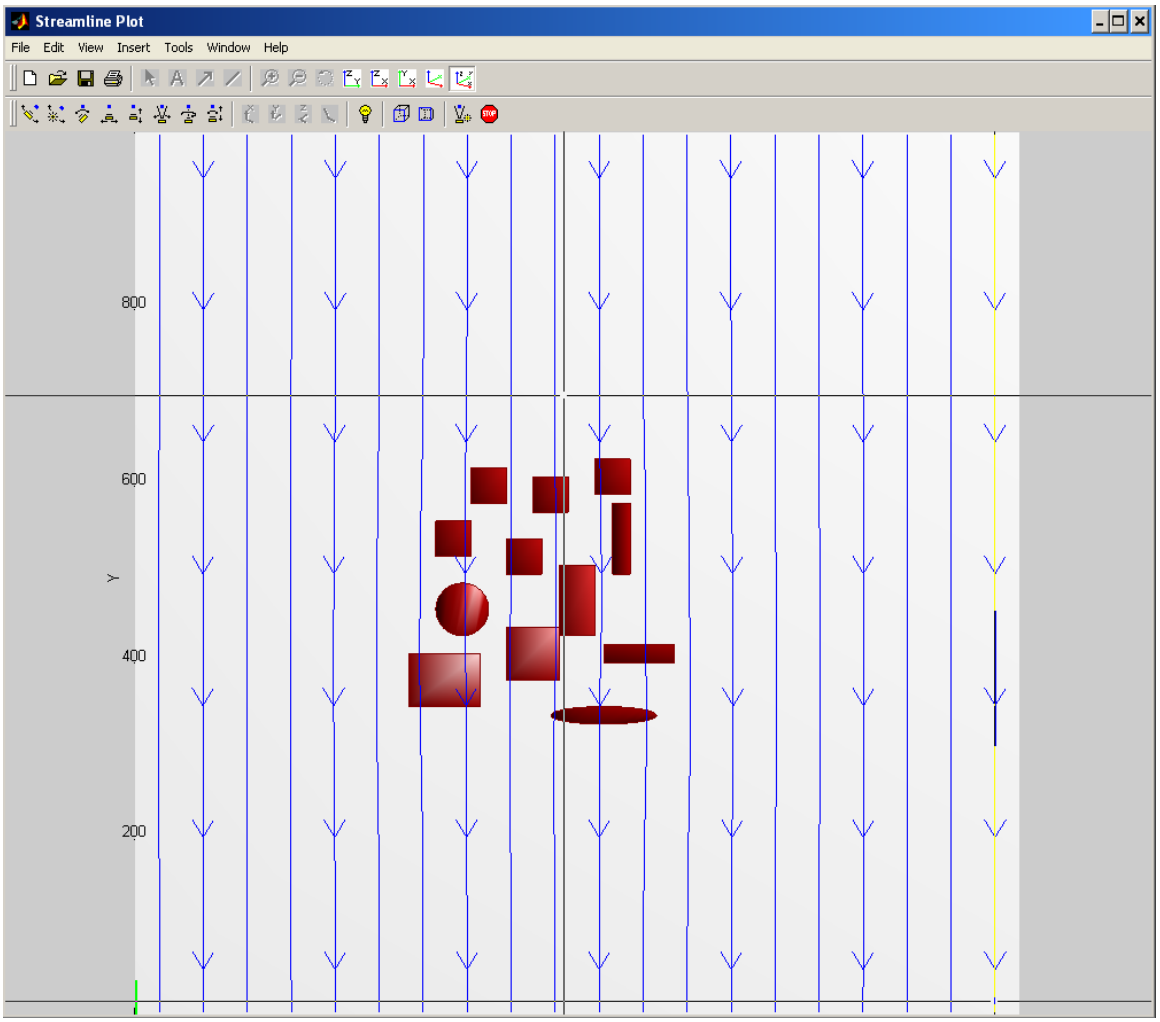



Choose the location of each particle by either entering the x, y and z co-ordinates of the location of choose the 'Select Release Pts' 

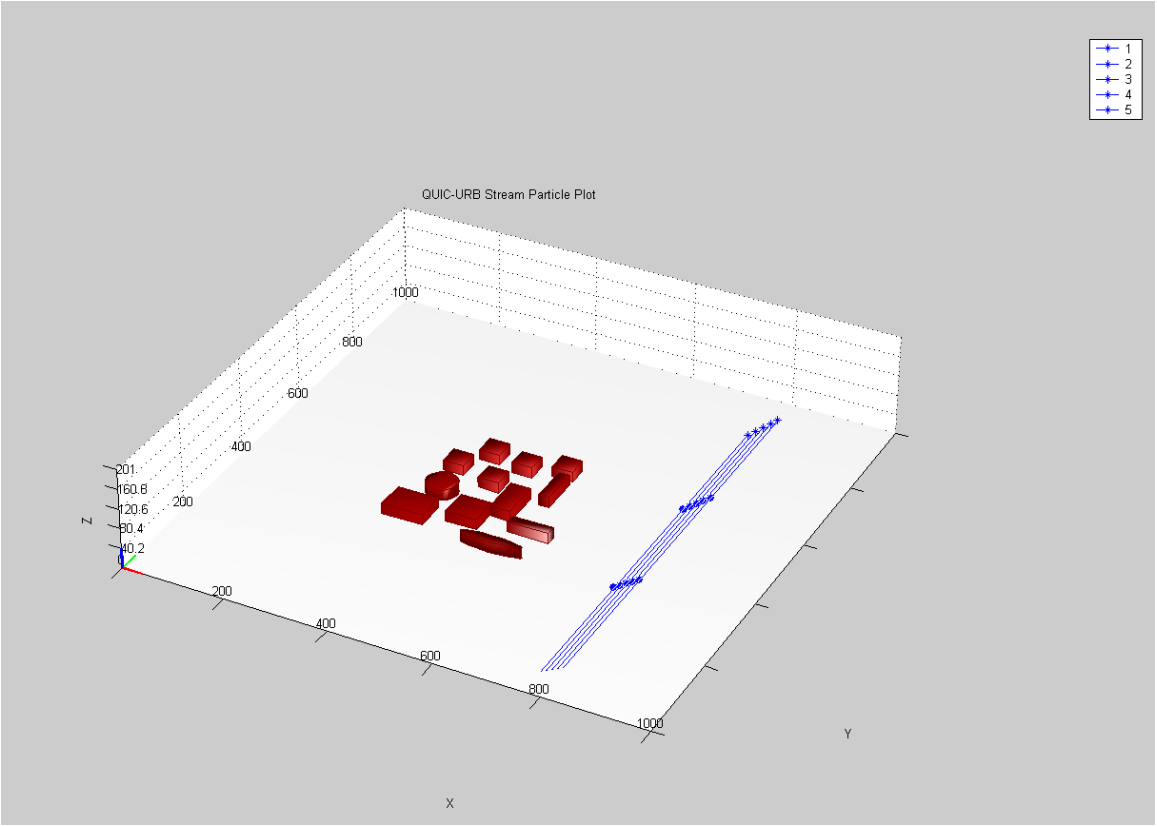
button

Clicking the 'Select Release Pts'

button brings up a window where the user can select the release points with the cursor.

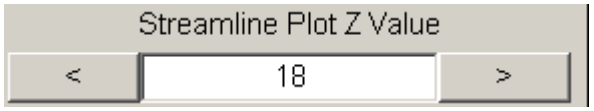



If entering the x, y and z coordinates, press the 'Plot Pathlines' button Plot Pathlines. This plots the pathlines at the specified locations.

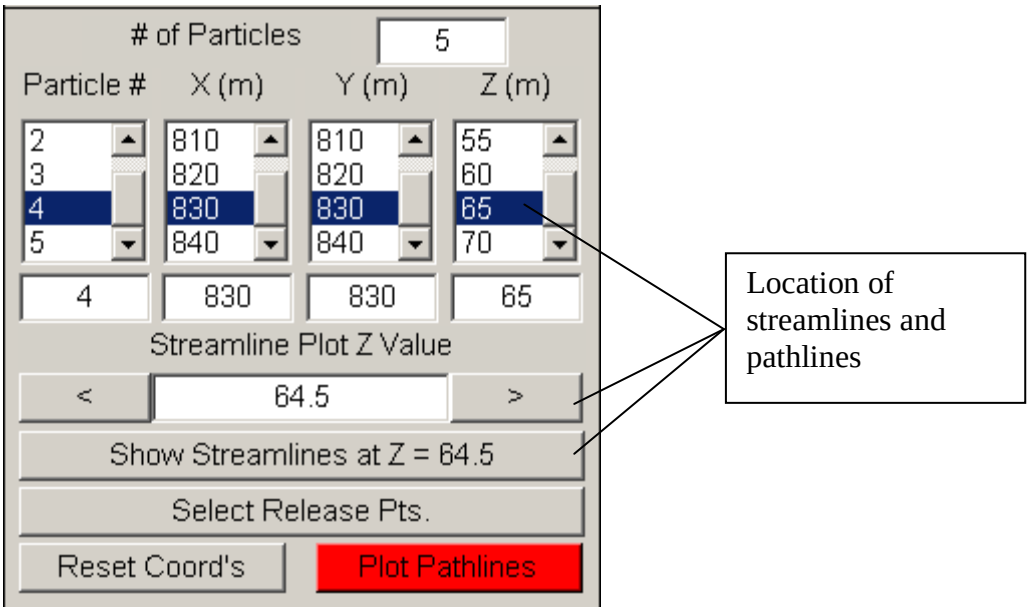



**Pathline Plot**

To view the streamlines, enter the value of the location of the steamlines in the 'Streamline Plot Z Value' window or use the 'Position location' button > to navigate to the desired location

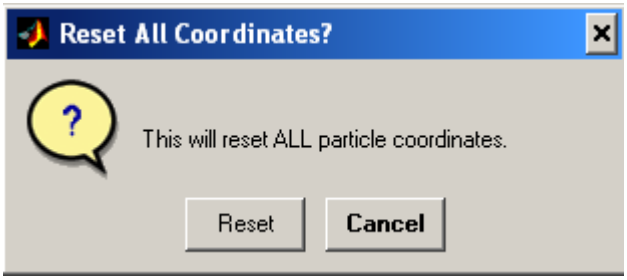



To view the streamlines at the same locations as that of the pathlines, select the location of the pathlines. This by default changes the location of the streamlines to that of the pathlines in the 'Streamline Plot Z Window' and the 'Show Streamlines at Z' button. This is shown below.




To reset the particle coordinates, click the 'Reset Coord's' button


This will bring up a pop up window as shown



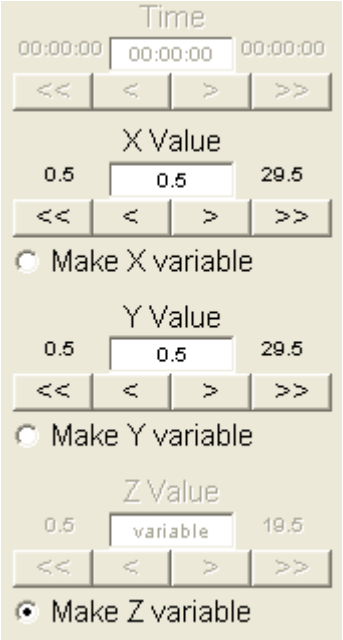

Click on 'Reset'

to reset the coordinates

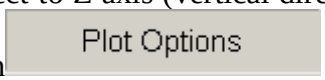
# Velocity Profiles

Velocity profiles can be viewed by pressing the ‘Velocity Profiles’ button..

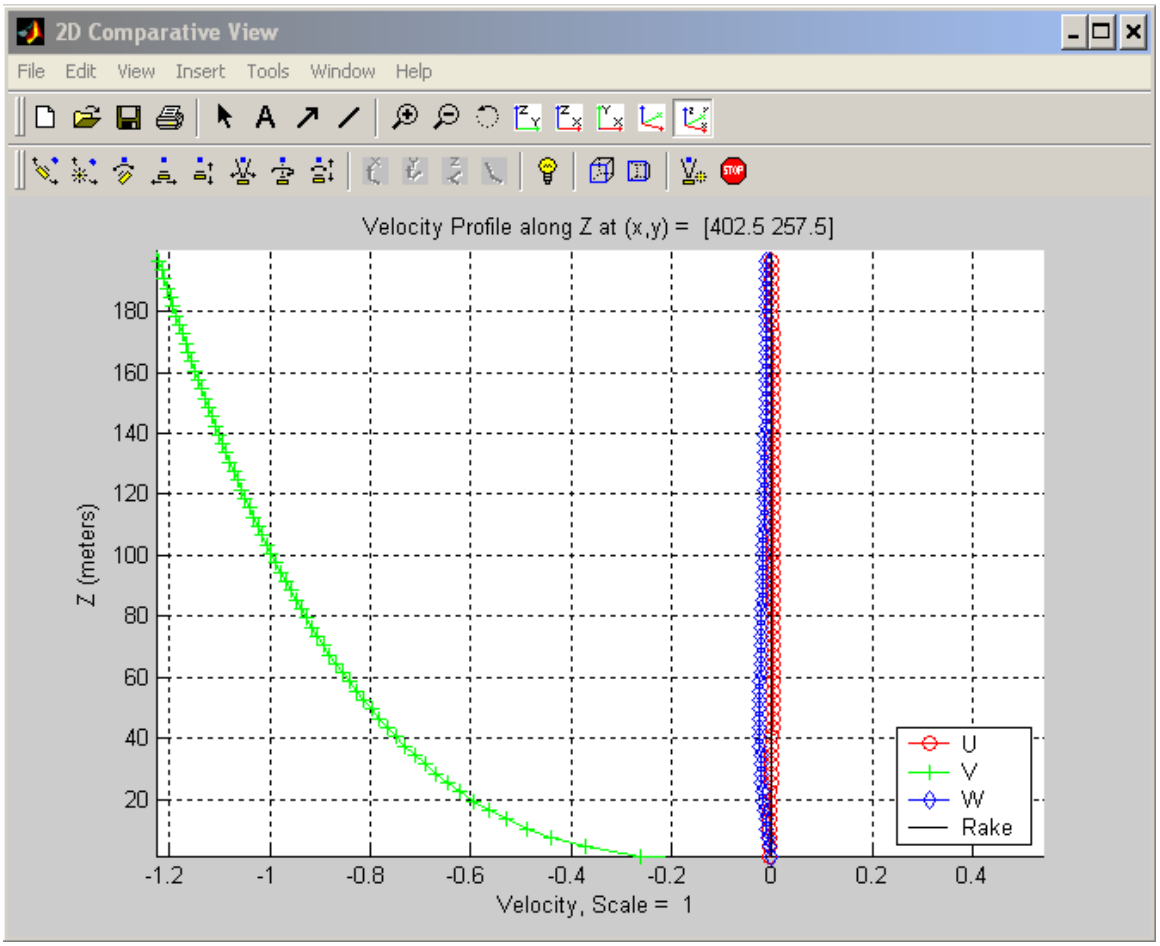
Line profiles at a particular location and along a particular axis can be viewed by selecting the particular axis. This is shown below.



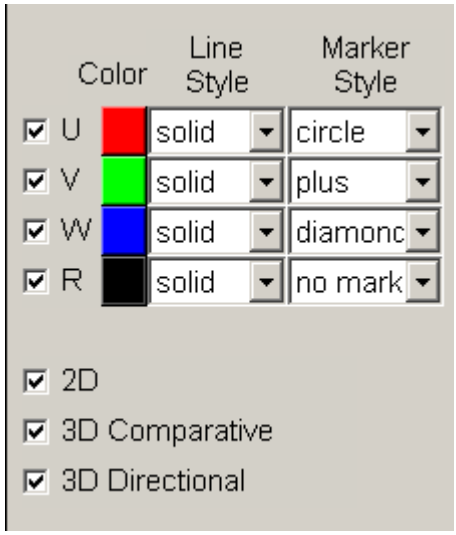

‘Making Z variable’ plots the velocity profiles with respect to Z axis (vertical direction).

For additional plot options, click the ‘Plot Options’ button 


The figure below shows the velocity profiles at a particular location.




The user can choose to plot any of the three velocity component at a particular location by selecting or deselecting the respective components. For the figure above, all three velocity components were plotted.



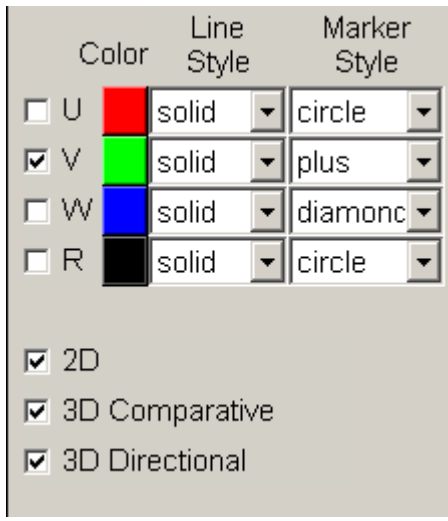
Deselecting all the three components and selecting only the Rake ('R') plots the location where the velocity profiles were intended to be plotted.

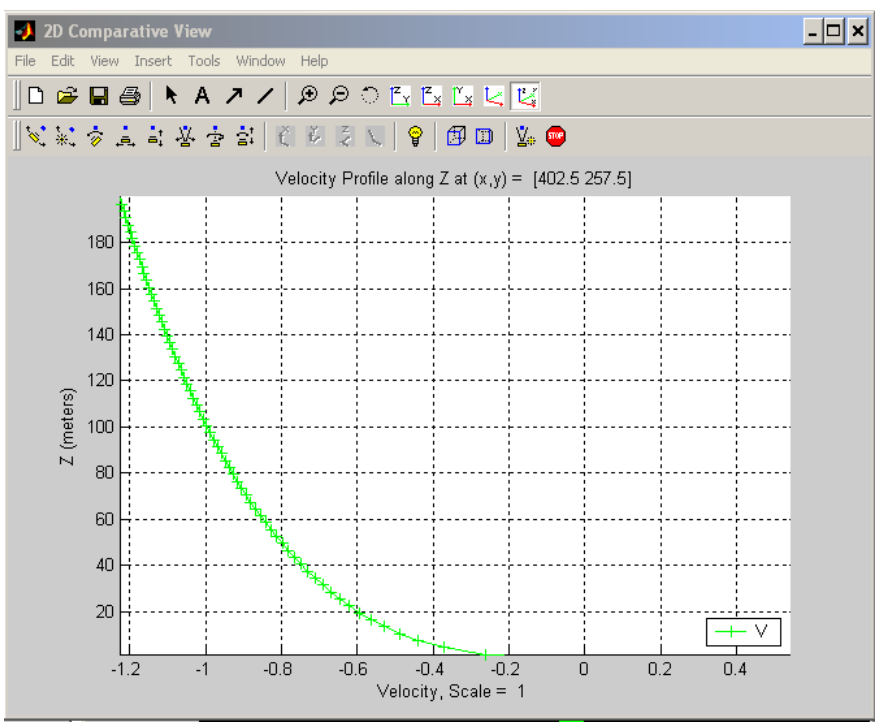
Selecting '2D' plots the velocity profiles on a 2D plane

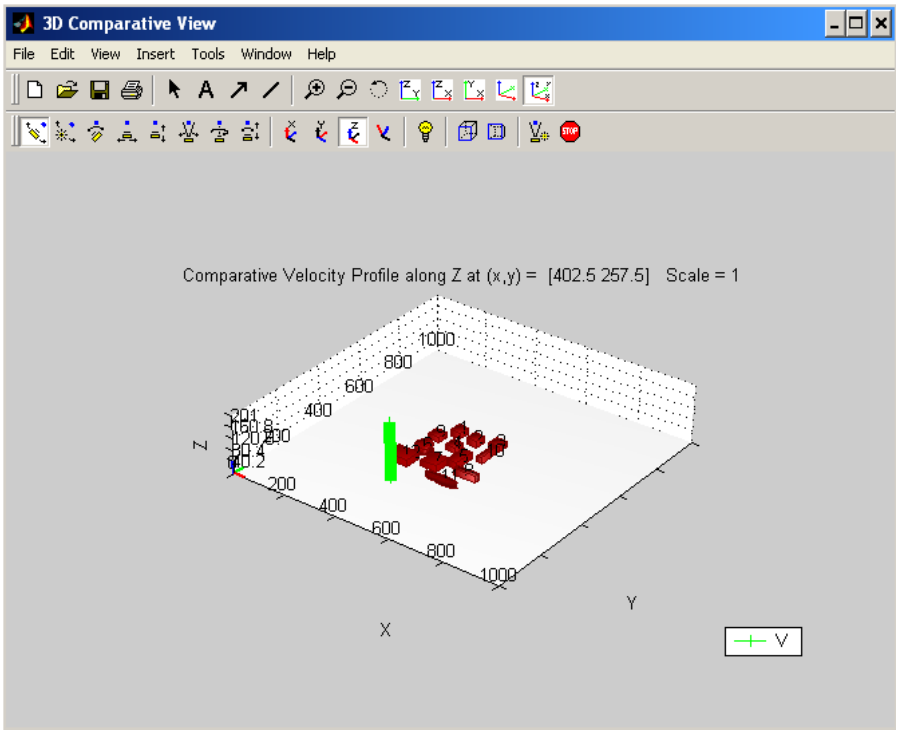
Selecting '3D Comparative' plots the velocity in the 3D domain

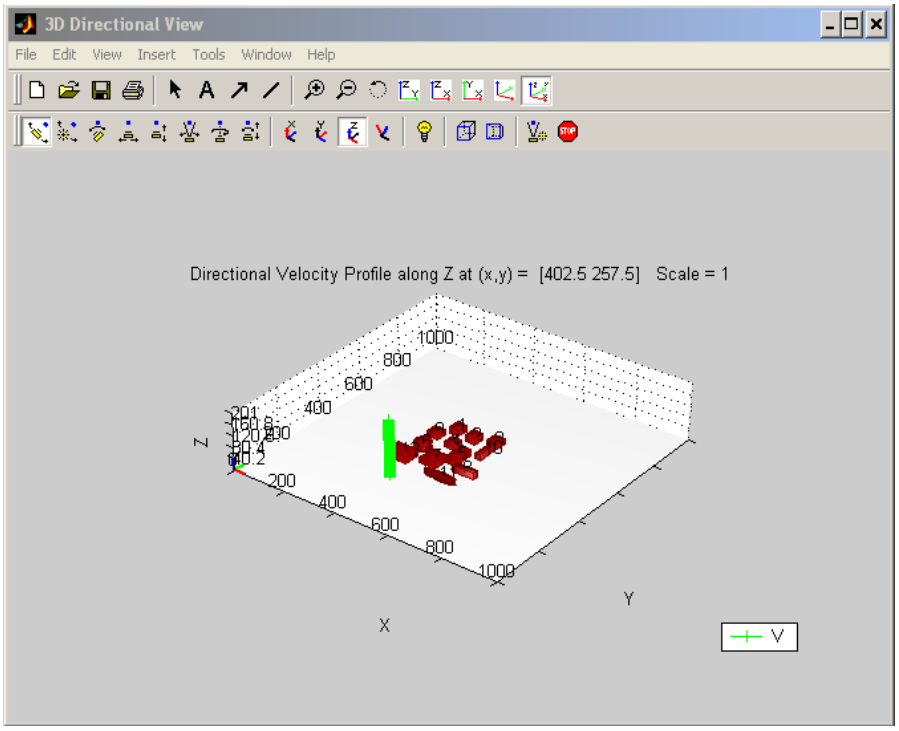
Selecting '3D Directional' gives the user the directional velocity profile

Consider the case where only the 'V' component is being plotted. For this case, the aforementioned plots are shown.



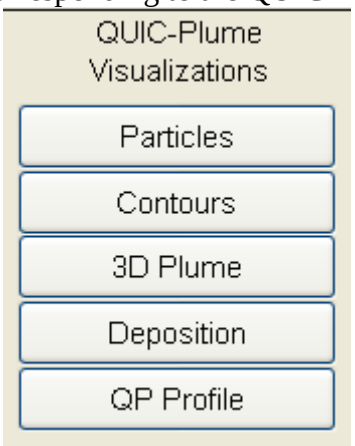




3D Comparative View
File Edit View Insert Tools Window Help
Comparative Velocity Profile along Z at (x,y) = [402.5 257.5] Scale = 1
1000
800
600
400
200
400
600
800
1000
Z
X
Y
V


3D Directional View
File Edit View Insert Tools Window Help
Directional Velocity Profile along Z at (x,y) = [402.5 257.5] Scale = 1
1000
800
600
400
200
400
600
800
1000
Z
X
Y
V

## QUIC PLUME VISUALIZATIONS

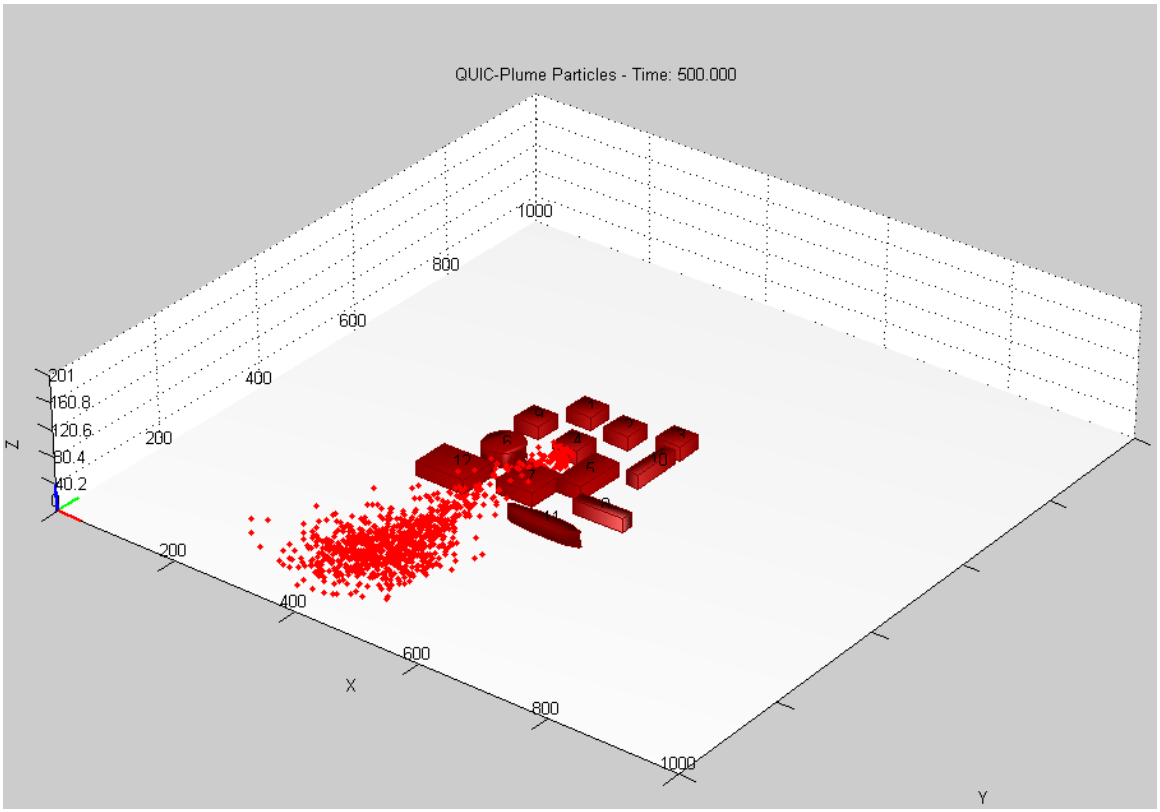
Below are the toggle buttons corresponding to the QUIC-PLUME data visualizations




### Particles

To view the locations of the particles released at a particular time step, press the 'Particles' button.

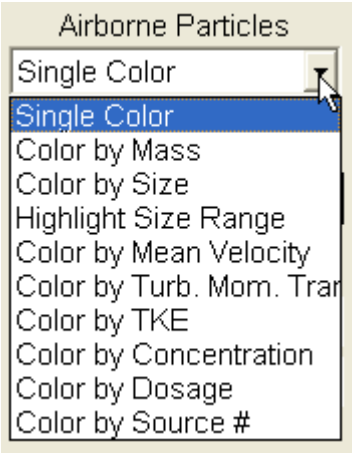
The user can select the time at which he / she wishes to see the particle positions by entering the value of time in the 'Time' window or by navigating using the 'Time Selection' buttons .

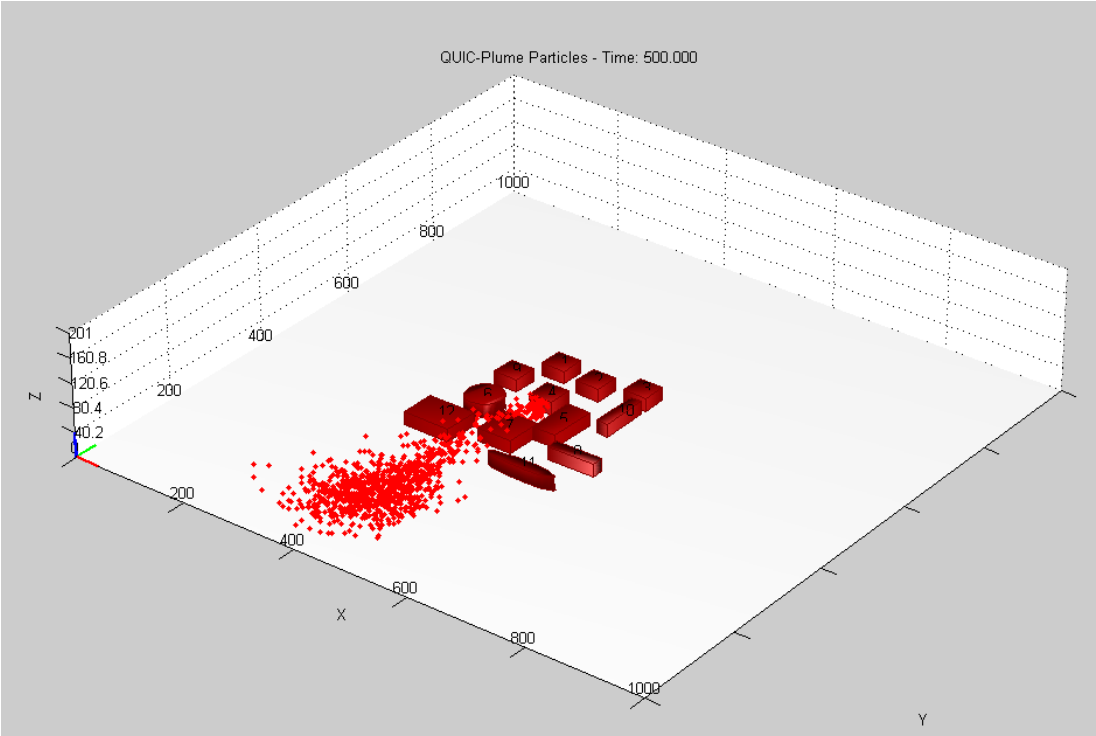



For additional plot options, click the 'Plot Options' button

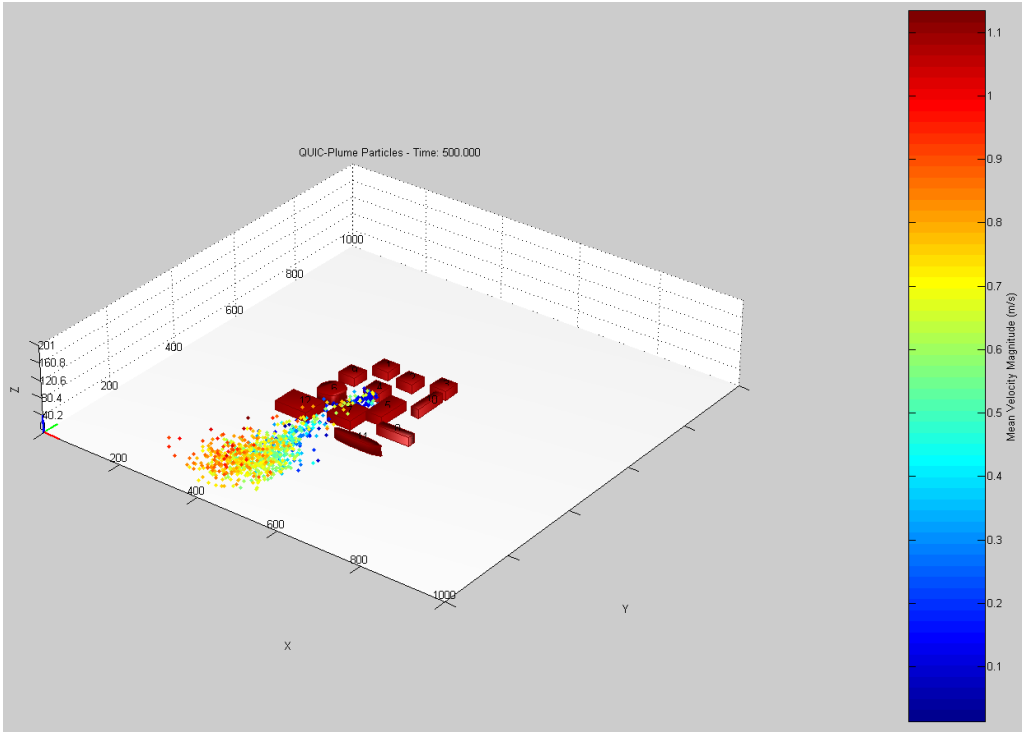
## *Coloring Airborne Particles*

The 'Airborne Particles' option provides the user of visualizing the particles based on various properties. This is illustrated through the figure below.

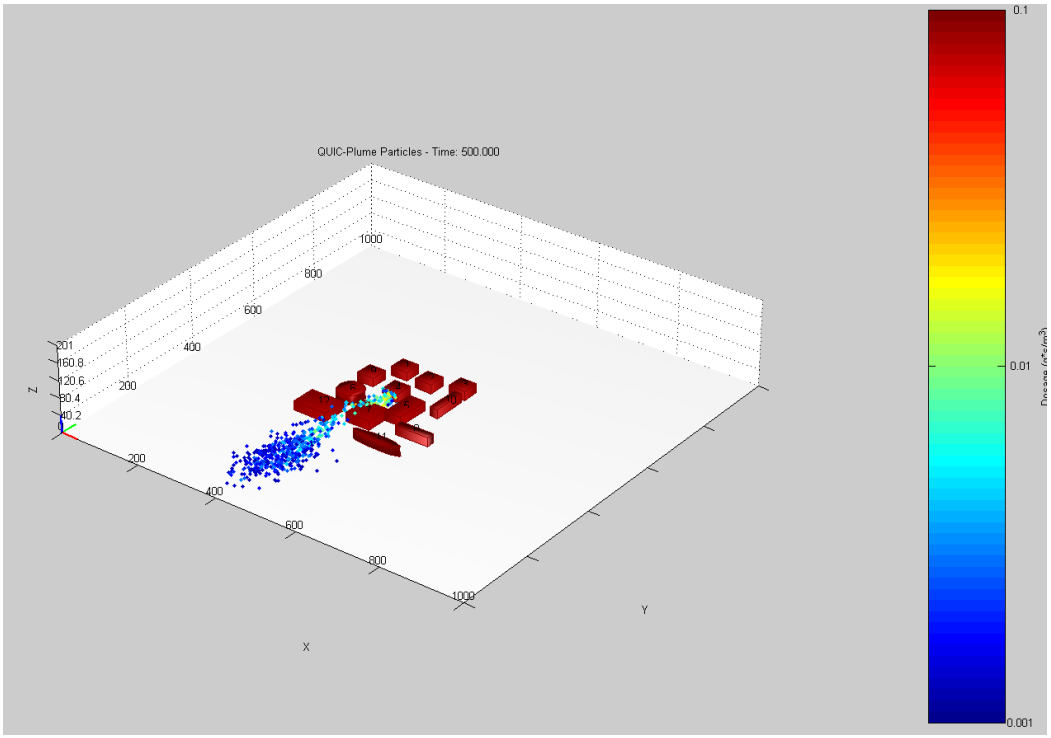



Single Color



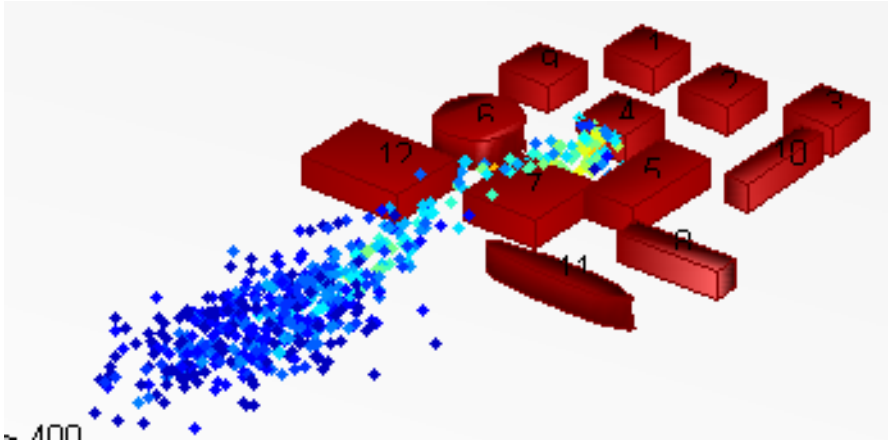

Color by Mean Velocity



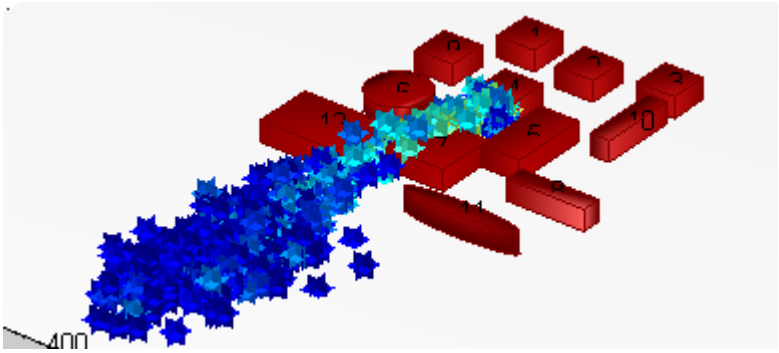

Color by Dosage

## *Quality*

The quality option if for enhancing the graphic effects of the plots. Choosing 'High' gives better graphical resolution but takes longer time.



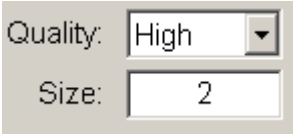
Quality - Low

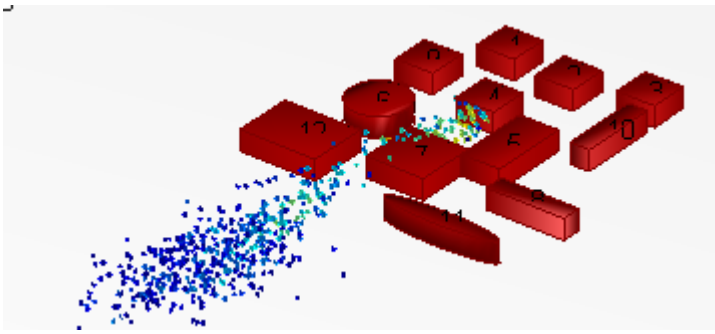


Quality – High

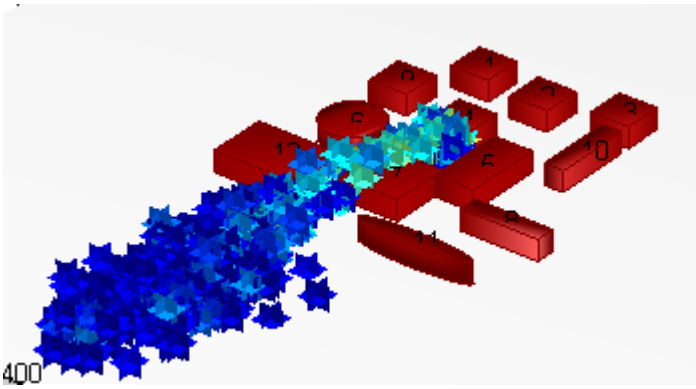
## *Size*

Size determines the size of the particles plotted.



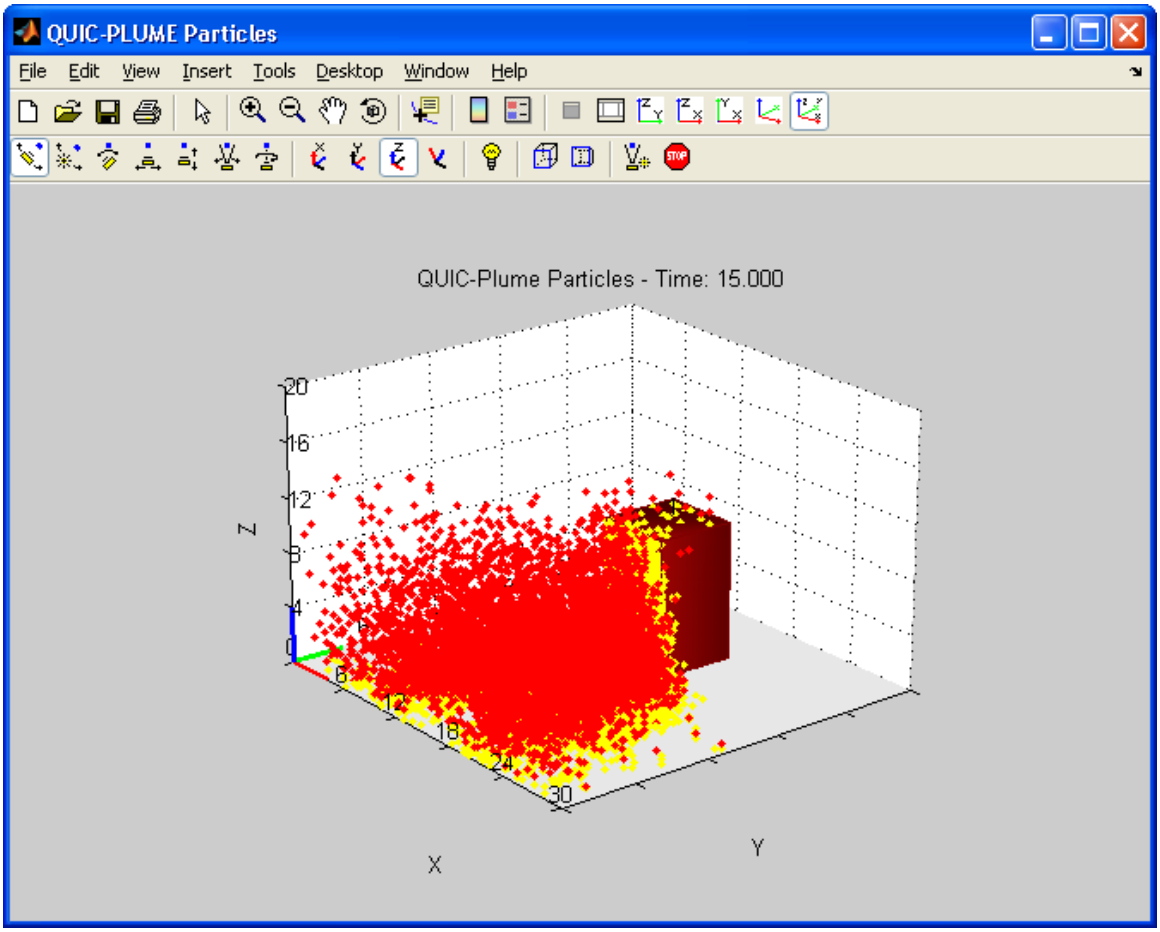
Size = 2




Size = 12

## *Deposited Particles*

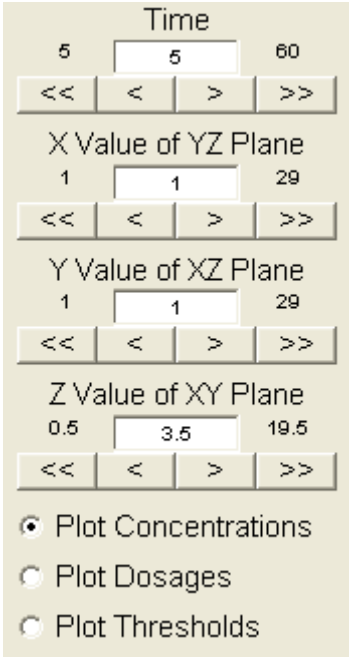
If the deposition velocity was not set to 0 in the Source Setup GUI, the deposited particles can be plotted with the airborne particles. Currently the deposited particles can only be plotted as a single color. In the plot below the yellow particles are the locations where material has been deposited on the surfaces and the red particles are airborne.

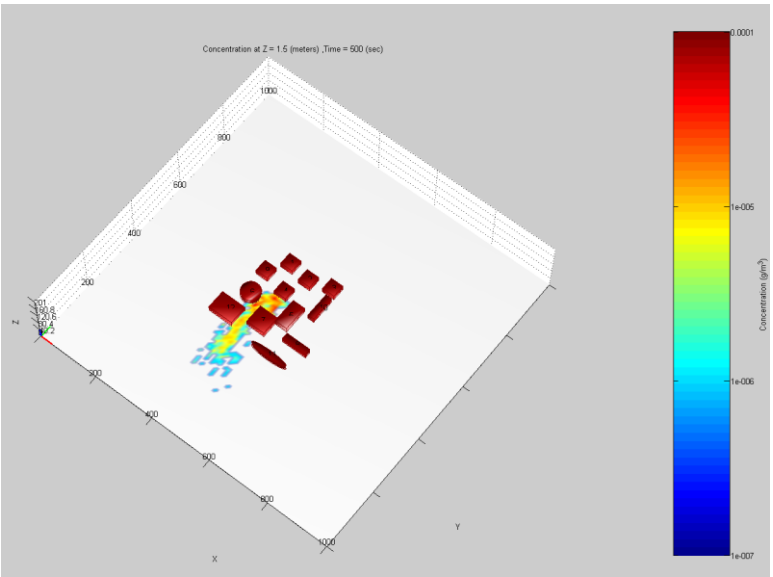


# Contours

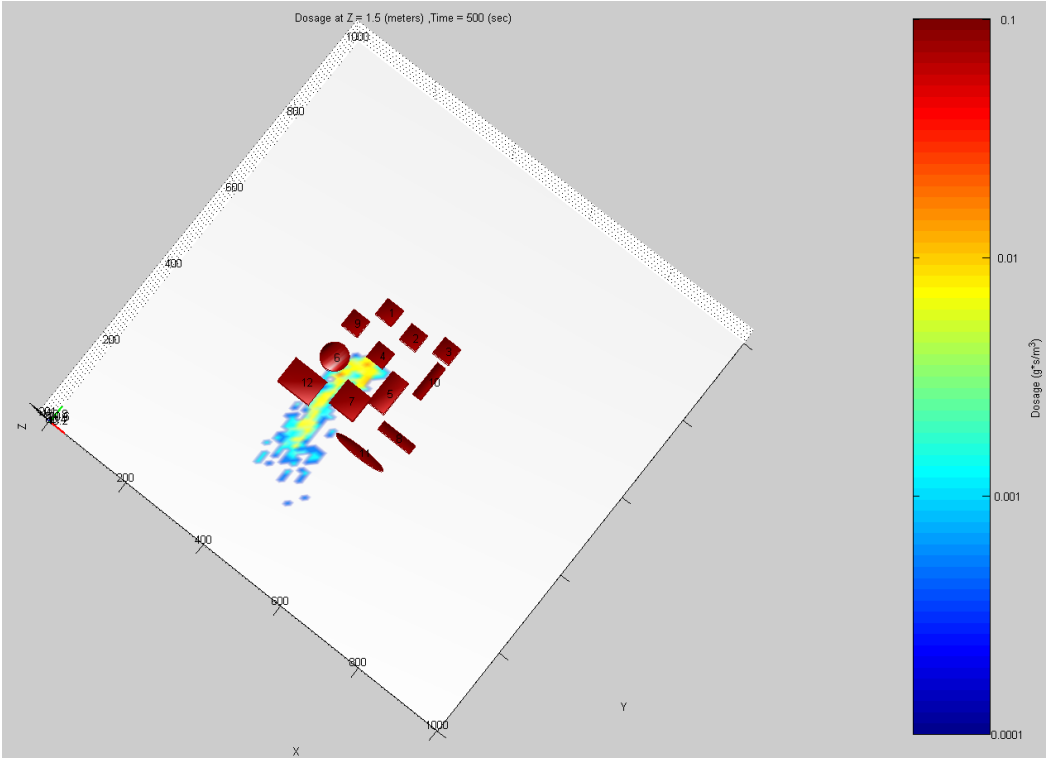
Results from QUIC-PLUME can be visualized as contour plots. To do so, click the 'Contours' button

Contours of Concentrations, Dosages and Thresholds can be plotted at different time intervals and along various planes by selecting or deselecting the various options shown below.

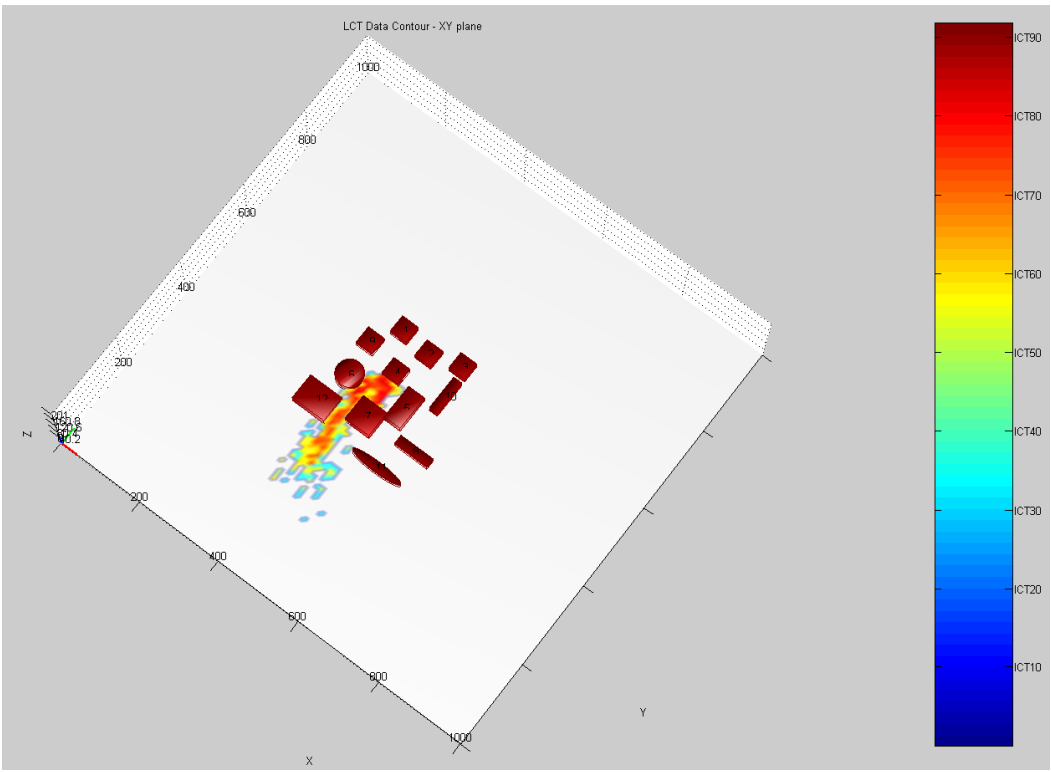


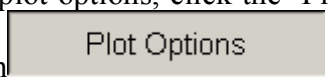

Contour of Concentration along the X-Y plane



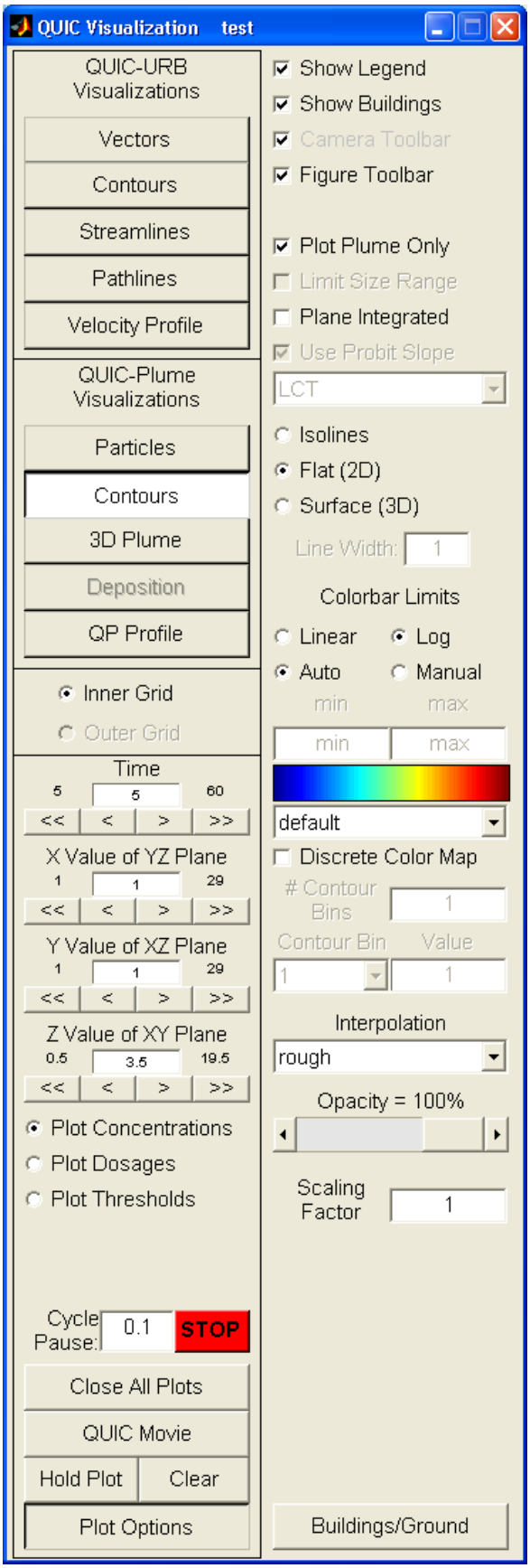

Contour of Dosage along the X-Y plane




Contour of LCT levels along the X-Y plane

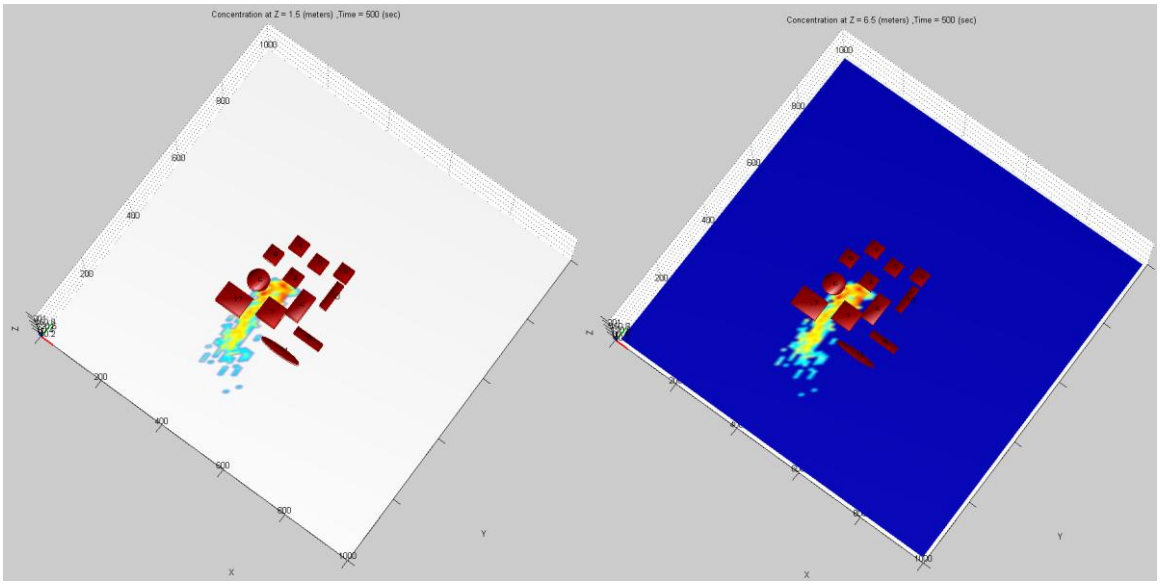
For additional plot options, click the 'Plot Options' button 


This brings up a window that has the additional plot options for viewing contour plots using QUIC PLUME. This is shown to the right.



## *Plot Plume Only*

Selecting the 'Plot Plume Only' plots the plume alone. Deselecting it adds a color to the surface. This is shown below.



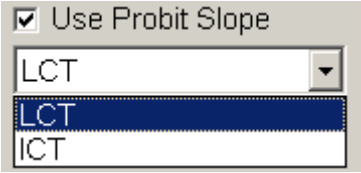

Plot Plume Only Deselecting Plot Plume Only

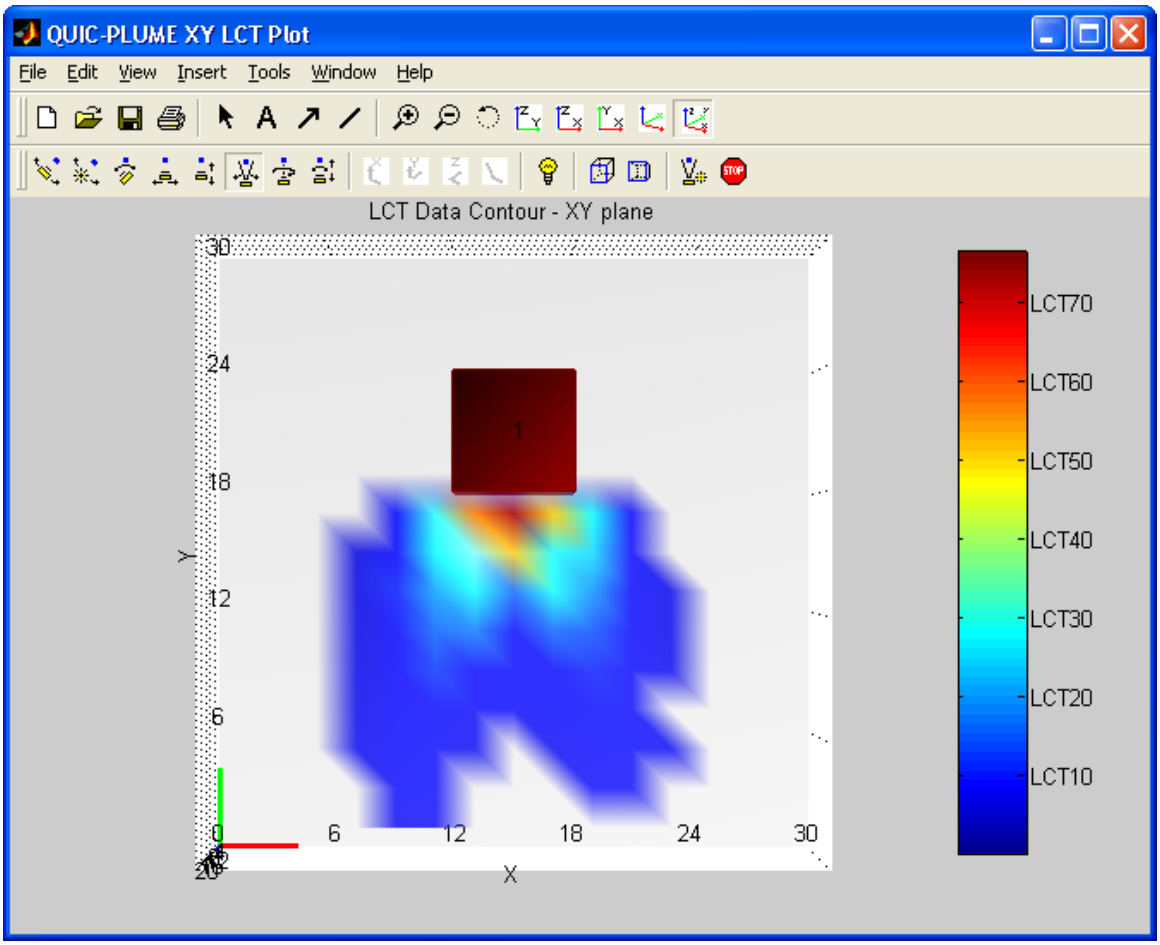
## *Plane Integrated*

Selecting the 'Plane Integrated' option integrates the contours of concentration or dosage or thresholds either laterally, longitudinally or vertically. That is to say that if the user wants to view the plane integrated concentration along the X-Y plane, it would be the result of all the concentrations integrated along the vertical planes (along all the columns).

## *Use Probit Slope*

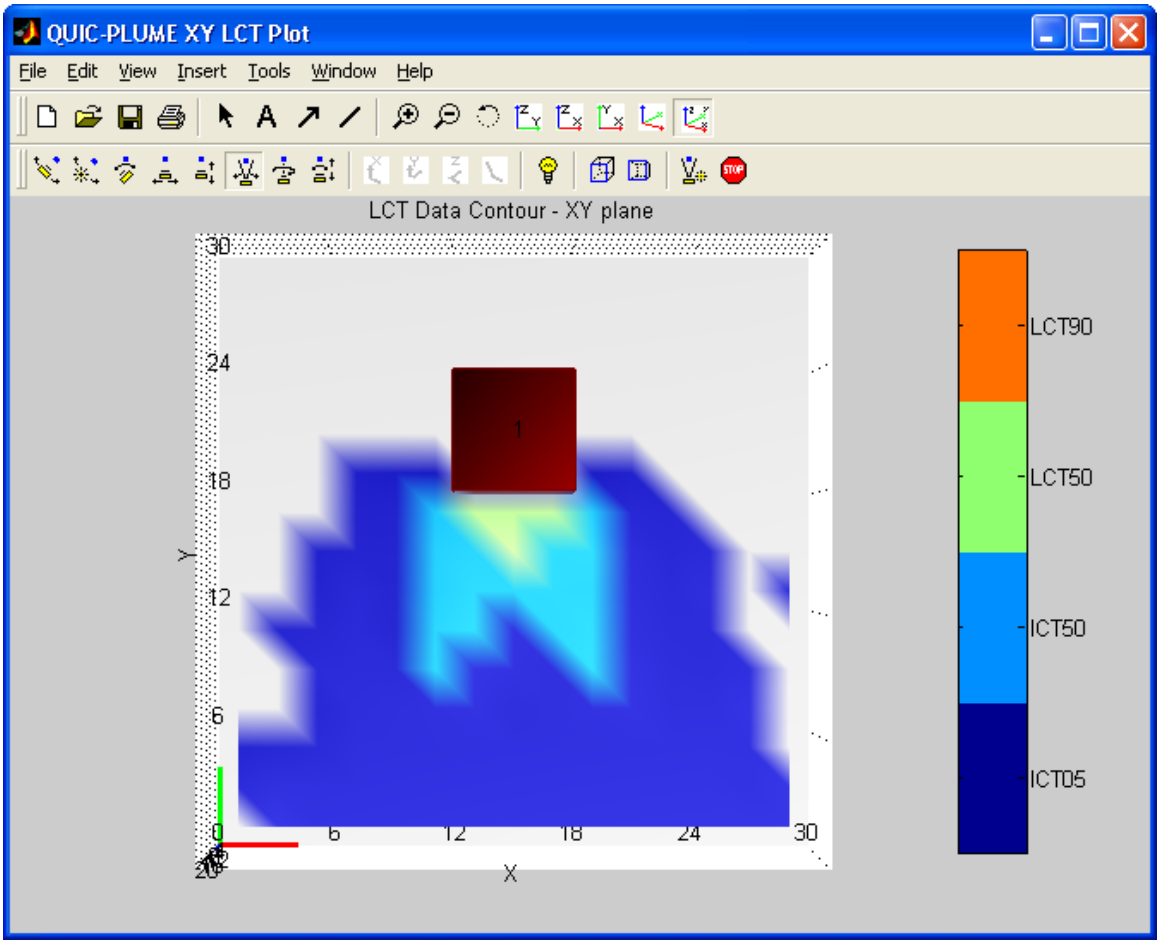
'Use Probit Slope' option works only with 'Plot Thresholds'. The probit slope method assumes a populations response to stimulus is linear in probit units (standard deviations from the mean population response). The actual type of response is arbitrary but two of the most common relating to chemical, biological, and radiological agents are lethal concentration thresholds (LCT's) and infectious or incapacitating concentration thresholds (ICT's). Several different population response curves can be defined for a single agent. The pop-up menu below the 'Use Probit Slope' check is used to select the desired population response curve for the plot as is shown below. The use of the probit slope method produces continuous contours of dosage responses. The levels correspond to percentages of the population that will exhibit the designated response after receiving the given dose.




LCT Data Contour – XY Plane using Probit Slope

When probit slope population response curves are not used the GUI uses the discrete population response contours that were set by the user in the Source Setup GUI. An example of the plot produced by the discrete population response contours is shown below.

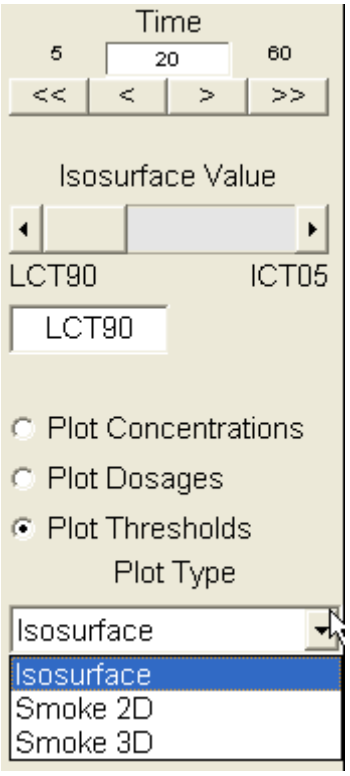



Discrete LCT and ICT Contours – XY Plane

The other plot options are similar to those described in the previous sections

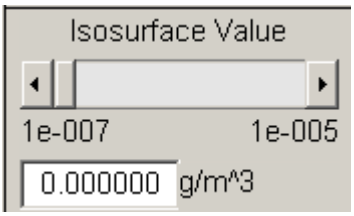
# 3D Plume Visualization

To view the plume in 3D, press the '3D Plume' button. 3D plume visualizations can be either isosurfaces, 2D textured smoke slices or 3D smoke volumes. The type of plot is selected using the popup menu and the type of data visualized is selected using the radio buttons seen below. Select a time at which the plume needs to be plotted by either entering the values manually or by using the time selection button

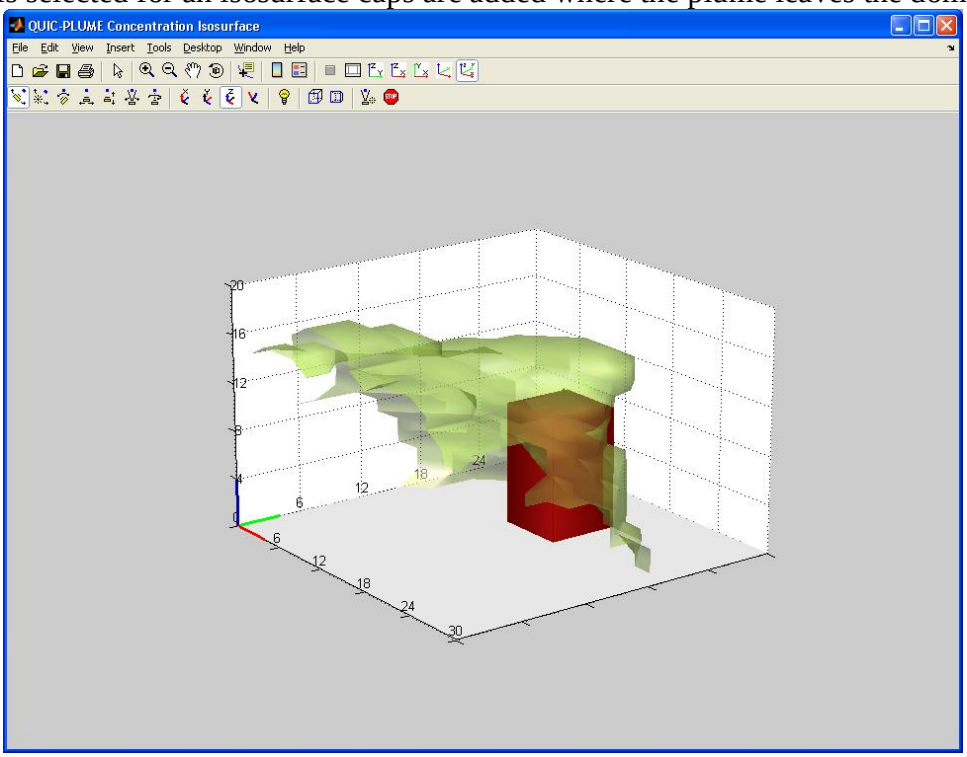



## *Isosurface*

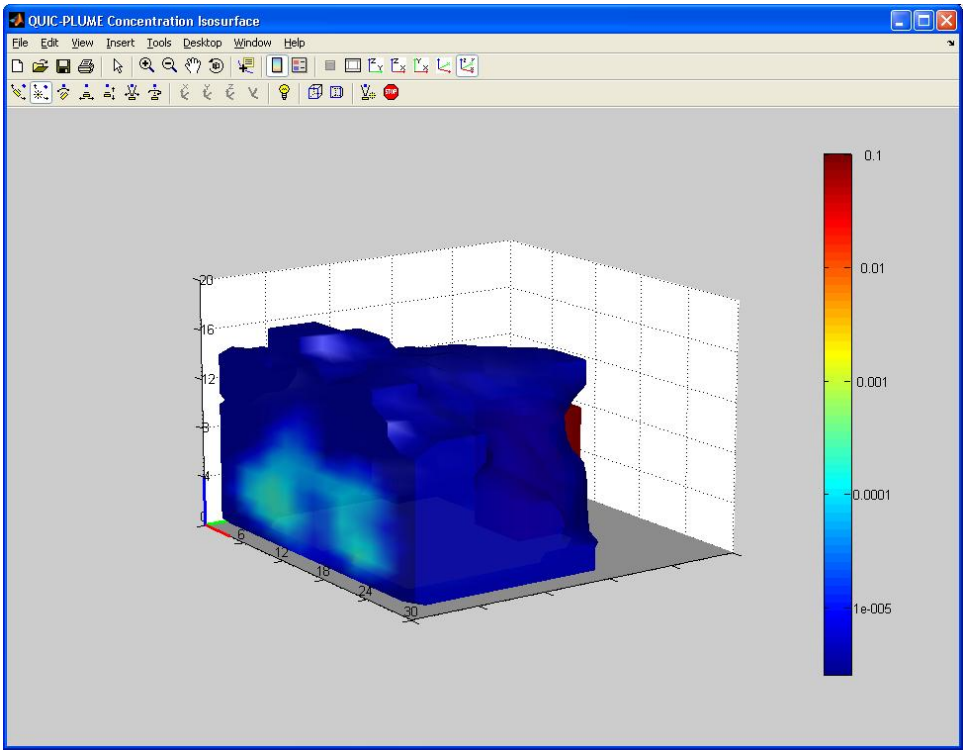
If the isosurface plot type is selected the isosurface value slider and edit box will be enabled.



For additional plot options, click the 'plot options' button. Similar to contours, the opacity and quality of the isosurface can adjusted. In addition, when the color map option is selected for an isosurface caps are added where the plume leaves the domain.



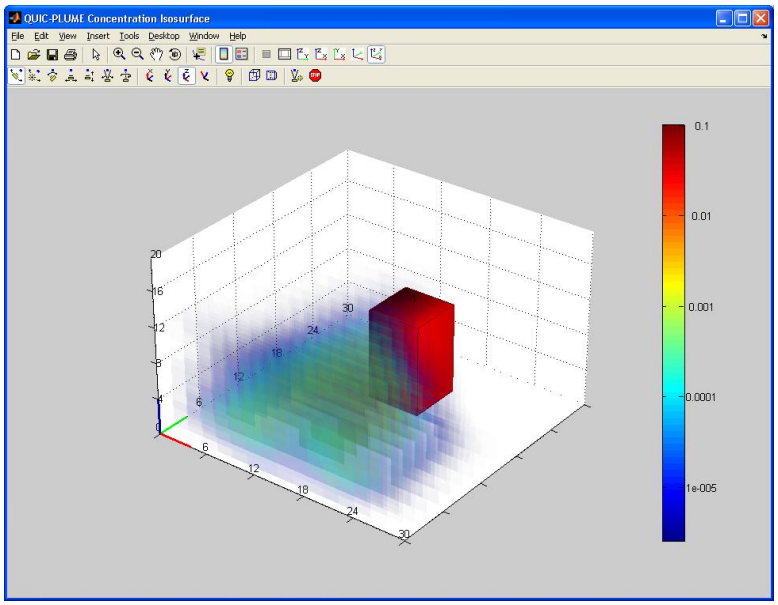

Single Color Isosurface



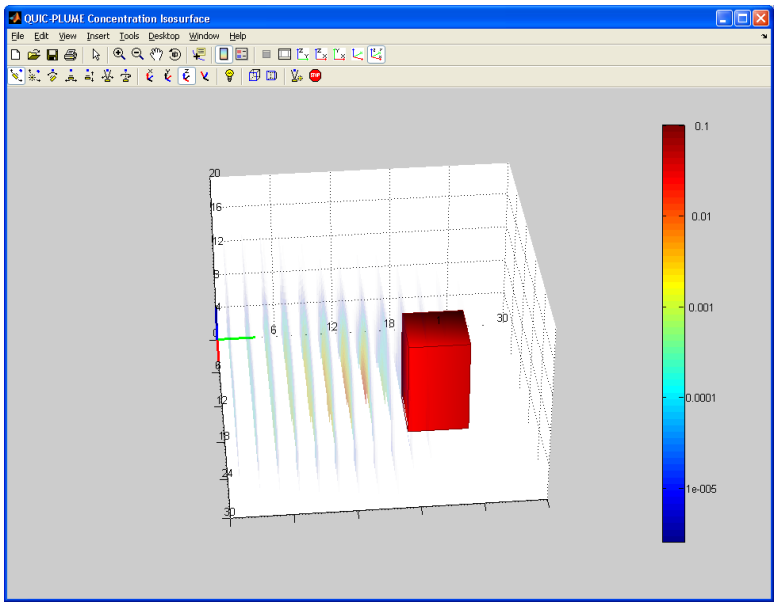

Color Mapped Isosurface with end caps

## *2D textured Smoke Slices*

The 2D textured smoke allows the user to see the entire plume at once rather than a single isosurface. The 2D textured smoke option produces color mapped slices of the plume. The code will chose the orientation of the slices for optimal visualization of the plume given the current orientation of the figure. If the plot is reoriented after plotting the 2D slices will be evident as seen below.



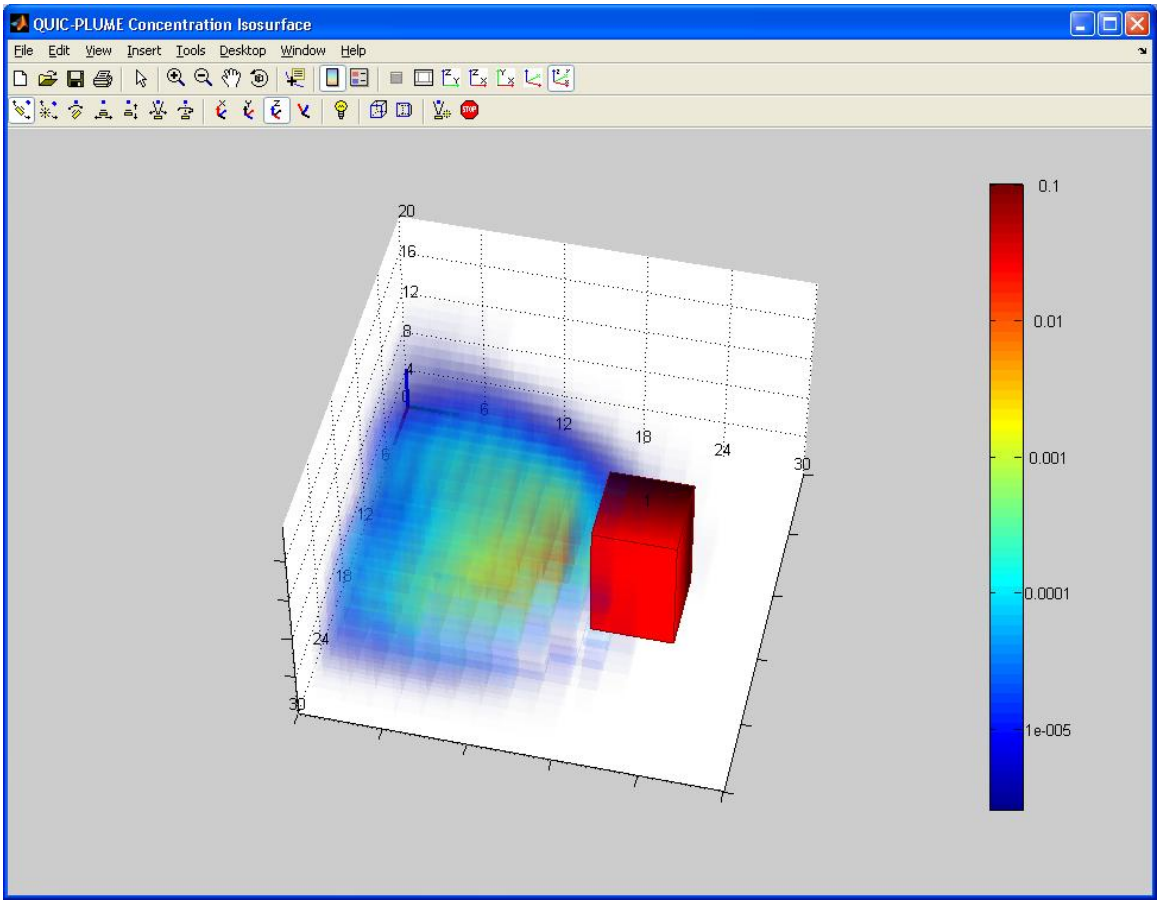

2D textured smoke plot before reorientation.




2D textured smoke plot reoriented to show the slices of the plume.

## *3D Textured Smoke Volumes*

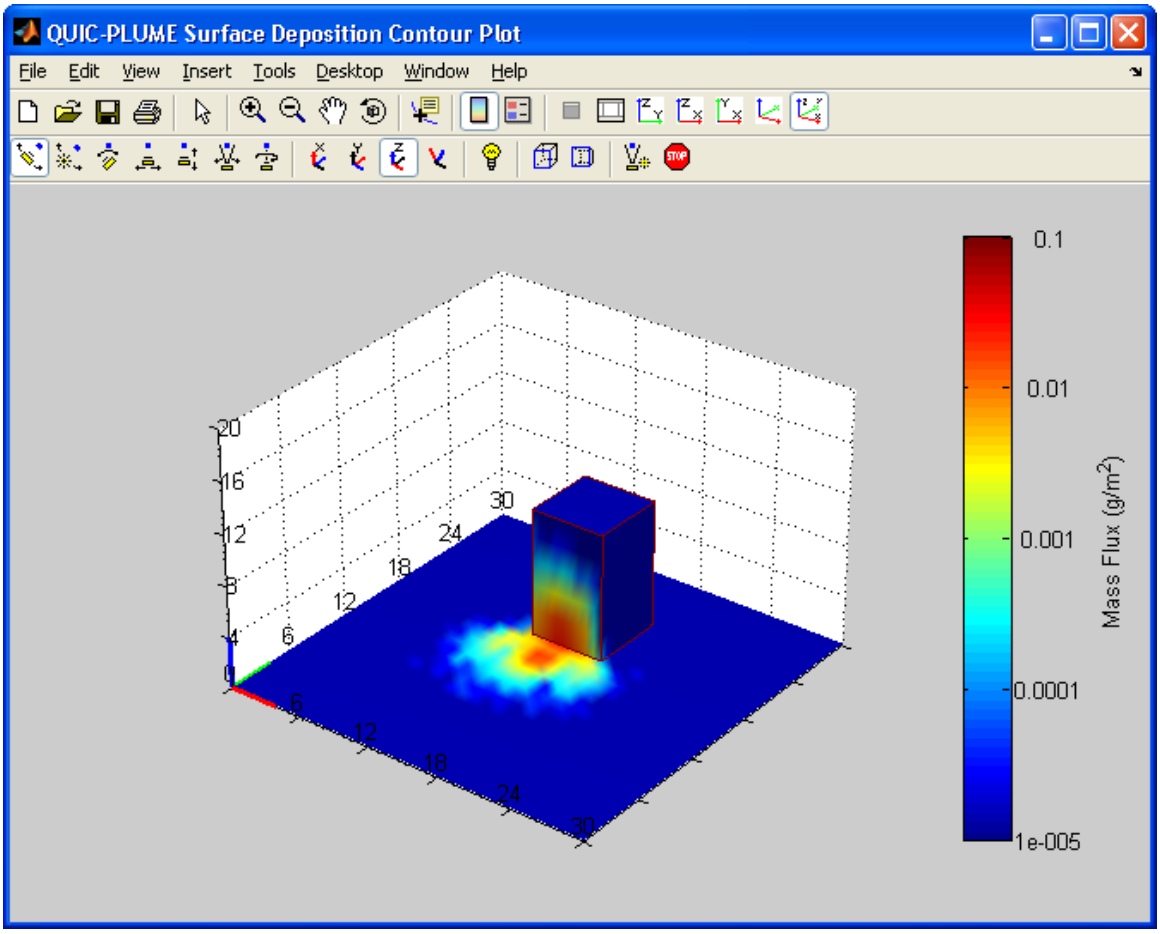
This option is very similar to the 2D textured smoke slices but it is not orientation dependent. It is, however, more computationally intensive and does not produce as vibrant of colors because there are more transparent patches used to produce the volumes.




3D Textured Smoke volumes of the plume.

# Surface Deposition

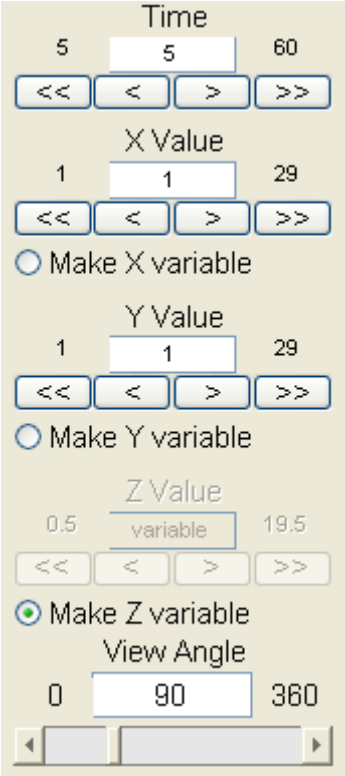
If the deposition velocity was not set to 0 in the Source Setup GUI then the surface deposition plot will be available. Similar to the airborne concentration/dosage fields, one can either plot the surface fluxes or thresholds. The thresholds can be discrete or can be continuous through the use of probit slope population response curves. An example of a surface deposition plot is shown below.

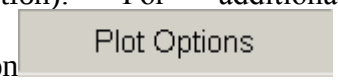


# QP Profile

QP Profile enables the users view the concentration profiles in 2D and 3D planes

Line profiles of concentration at a particular location and along a particular axis can be viewed by selecting the particular axis. This is shown below.




'Making Z variable' plots the concentration profile with respect to Z axis (vertical direction). For additional plot options, click the 'Plot Options' button 

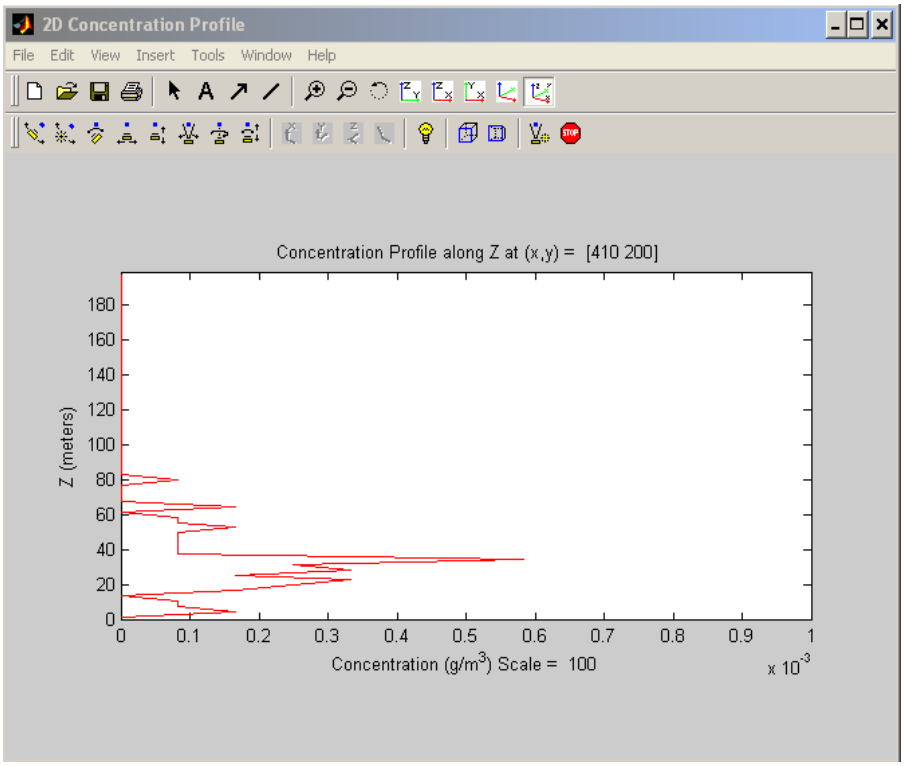

Selecting '2D Profile Plot'

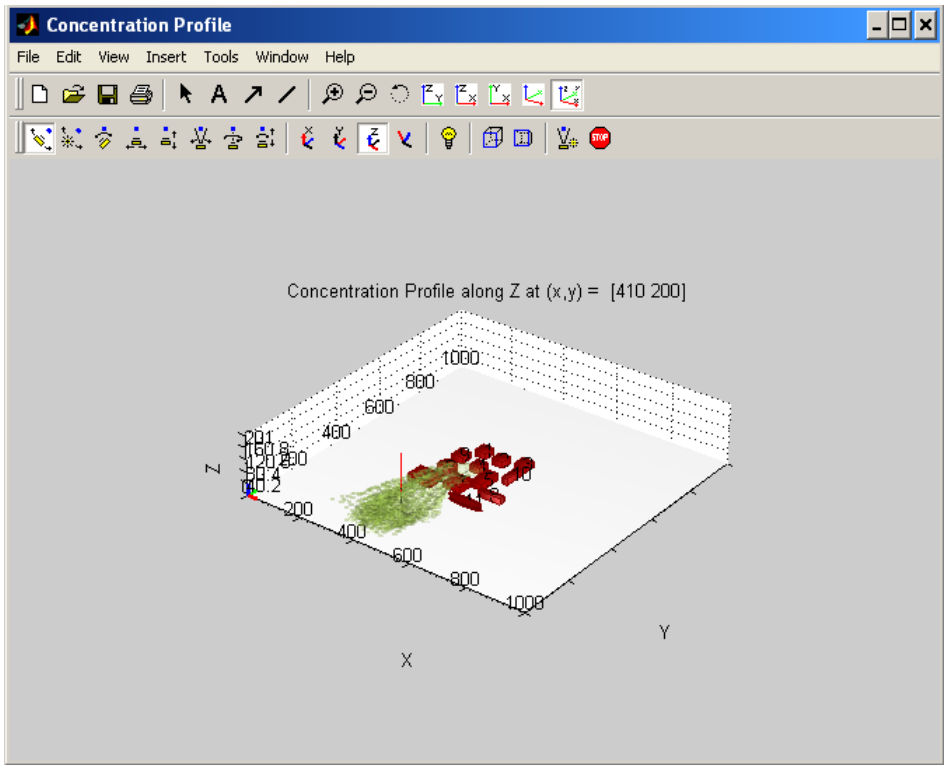
plots the concentration profile in the 2D plane

Selecting '3D Profile Plot'

plots the concentration profile in the 2D plane

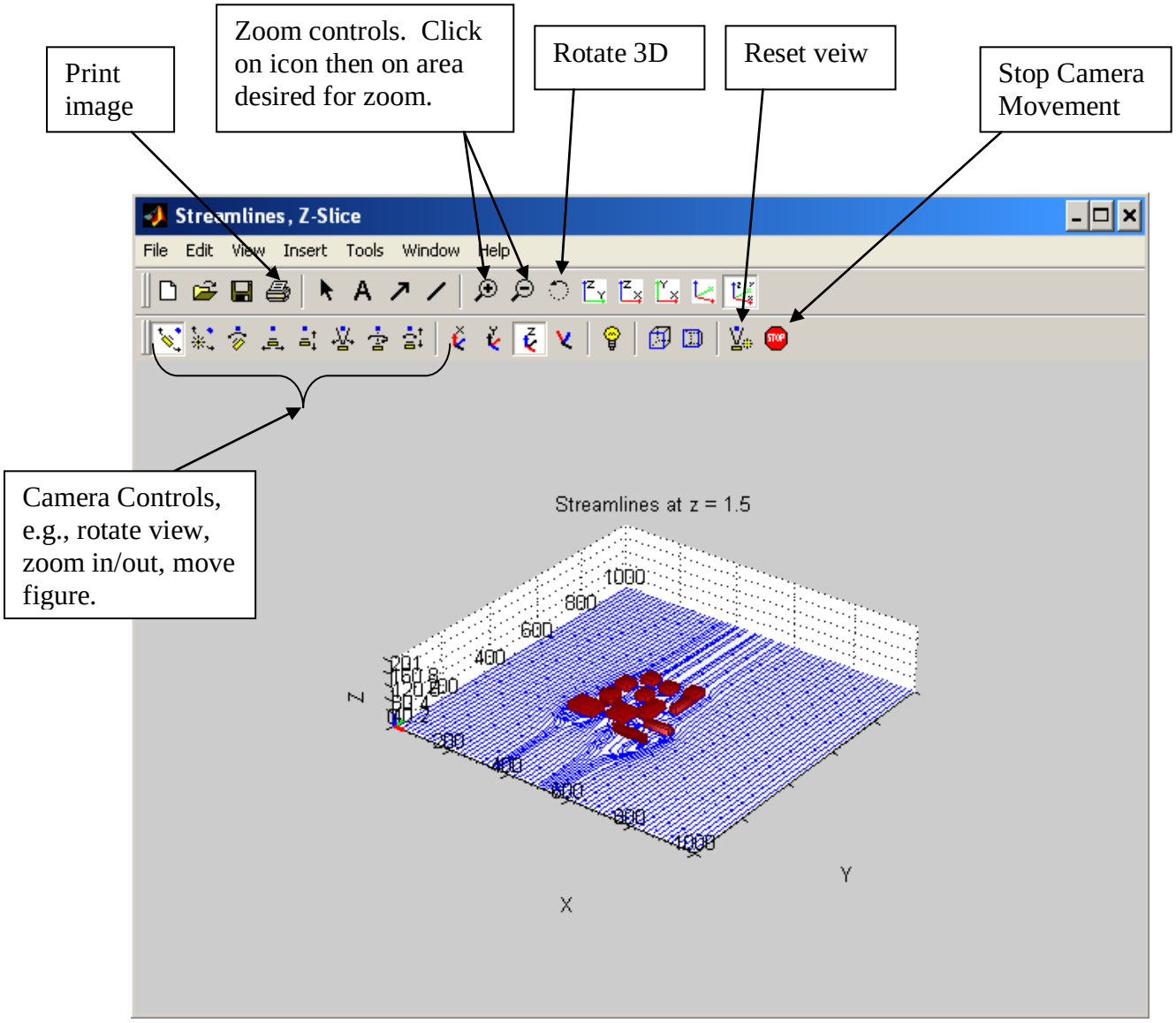
This is shown below


2D Concentration Profile
File Edit View Insert Tools Window Help
Concentration Profile along Z at (x,y) = [410 200]
Z (meters)
180
160
140
120
100
80
60
40
20
0
0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1
Concentration (g/m3) Scale = 100
x 10-3

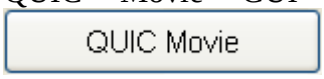

Concentration Profile
File Edit View Insert Tools Window Help
Concentration Profile along Z at (x,y) = [410 200]
Z
X
Y

## Plot controls

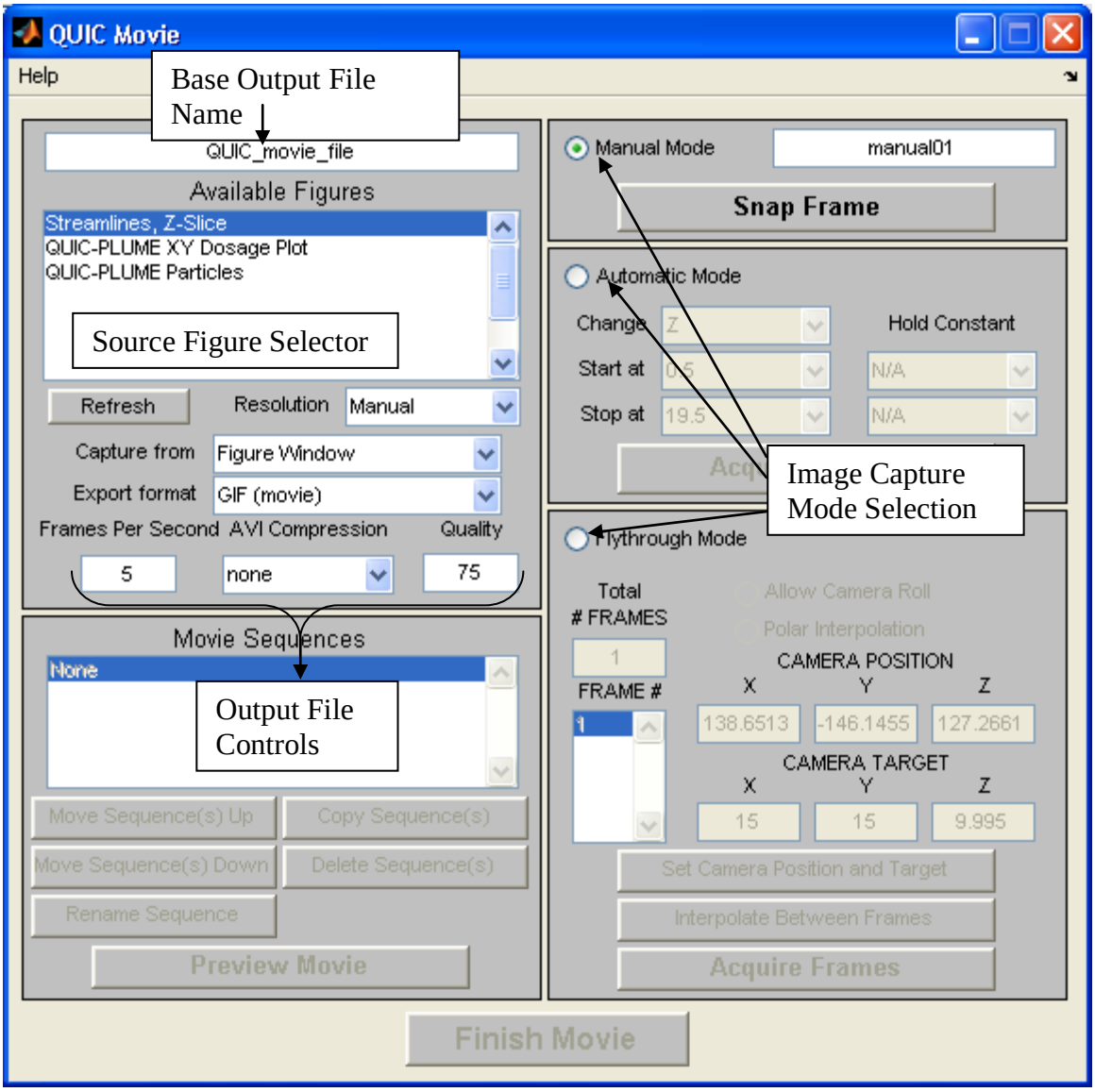
The example streamline figure below shows the different options for manipulating the plot. The controls are standard Matlab features.



# QUIC Movie GUI

QUIC Movie GUI is accessed by pressing the QUIC Movie button 

at the bottom of the Visualization GUI. This GUI can be used to save animations of the graphical output of the QUIC Visualization GUI as animated GIF files, AVI movie files or numbered .jpg, .png, .bmp, or .tif files. The QUIC Movie GUI is shown below.

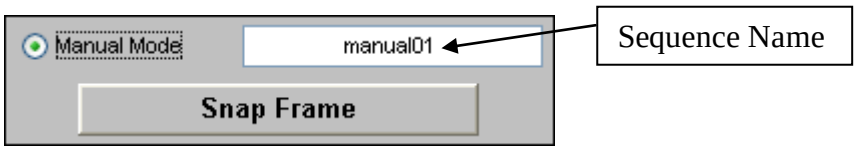


## Image Capture Modes

There are three basic modes for capturing animations of QUIC data plots produced by the Visualization GUI: Manual, Automatic, and Flythrough. Each of these modes will be discussed in detail below.

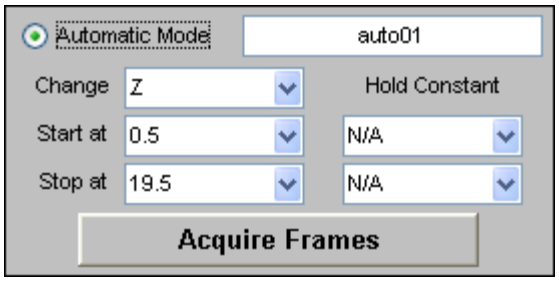
### Manual Mode

Pressing the 'Snap Frame' button in Manual mode simply captures a single image from the figure selected in Source Figure list box and adds it to the image sequence whose name is in the edit box above the 'Snap Frame' button. Editing the name of the current sequence will cause any subsequent image captures to be stored in a new sequence with the new sequence name. In manual mode all modifications to the image are performed outside of the Movie GUI.




### Automatic Mode

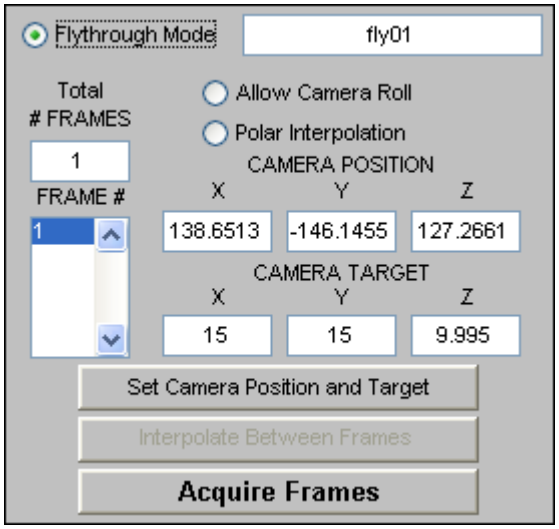
Automatic mode can be used to automatically capture images while cycling through the data (such as changing time or position). When the automatic capture mode is selected the automatic controls shown below become active.




Some plot types have two variables that can be modified such as contour plots which may have multiple time steps and plane positions. Other plots have only a single variable that can be changed such as particle plots where time is the only variable. Only one variable may be cycled through at a time. If two variables are available for the selected plot type, select the variable to cycle through with the pop-up on the upper-left hand side of the automatic controls. Then select the start and end values for that variable and select the value of the variable that is to be held constant. After setting all of the controls, initiate the frame capturing process by pressing the 'Acquire Frames' button. The plot will automatically cycle through the selected values and add the acquired frames to the sequence name found in the edit box.

# Flythrough Mode

The perspective of the 3D plots is defined by the camera position (the location of the observer) and camera target (the point which corresponds to the center of the figure). Flythrough mode allows the user to automatically modify the camera position and target over a user-specified number of frames in order to simulate flying through the 3D model. The flythrough controls are shown below.

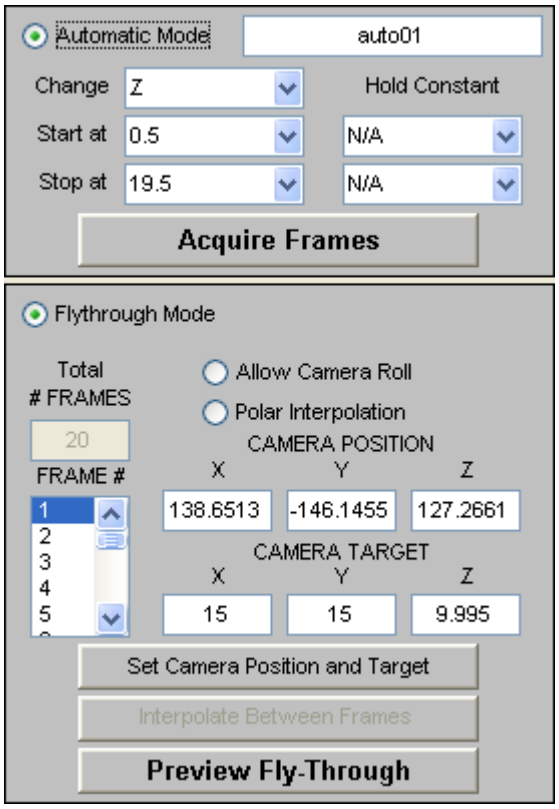



A flythrough frame sequence is produced by setting the total number of frames for the sequence in the edit box in the upper-left hand corner. The path of the camera position and target can either be set manually for each frame or can automatically be interpolated between beginning and ending locations. The camera position and target can be set manually set with the edit boxes or can be captured from the figure itself using the 'Set Camera Position and Target' button. Select a frame to use as the starting position using the frame number list box on the left. Manipulate the camera target and position using the camera controls found at the top of the figure window and save the current camera target and position by pressing the 'Set Camera Position and Target' button. Select an ending frame with the frame number list box and manipulate the camera position and target with the camera controls to the ending position and press the 'Set Camera Position and Target' button again to save the end location. Then select the frames to interpolate between by holding the 'Ctrl' button on the keyboard and left-clicking on the beginning and ending frames in the frame number list box. Once the camera target and position for the beginning and ending frames have been set and the frames have been selected in the list box, press the 'Interpolate Between Frames' button to automatically interpolate between the beginning and ending positions in the frames between the two selected frames. The interpolation defaults to being linear in Cartesian position between the points but selecting the 'Polar interpolation' radio button will cause the interpolation to be linear in radius and angle of the camera position relative to the target. The upwards vector of the camera position defaults to being parallel with the z-axis but this can be modified by activating the 'Allow Camera Roll' radio button. After the camera position

and target paths have been defined press the 'Acquire Frames' button to capture the frames and add them to the sequence name in the edit box at the top of the controls.

## Automatic Mode with Flythrough

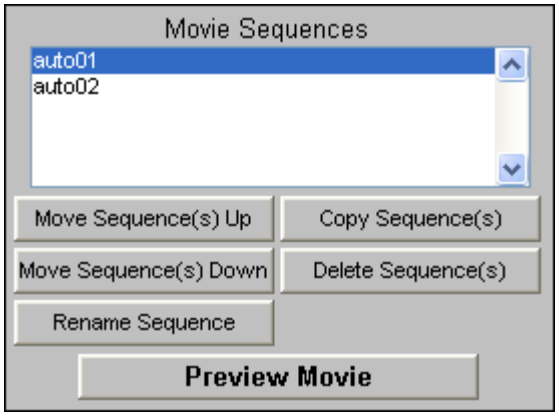
The automatic and flythrough frame capture modes can be used at the same time to cycle through the plot variables and change perspective at the same time. When both automatic and flythrough modes are selected the controls appear as shown below.




The flythrough controls work in the exact same way but the number of frames is determined by the start and stop values of the variable that is to be cycled through. The 'Preview Fly-Through' button can be used to check the paths of the camera position and target without capturing the images and saving them to the sequence in the edit box.

## Sequence Control

After at least one sequence has been captured the sequence controls become active as shown below.



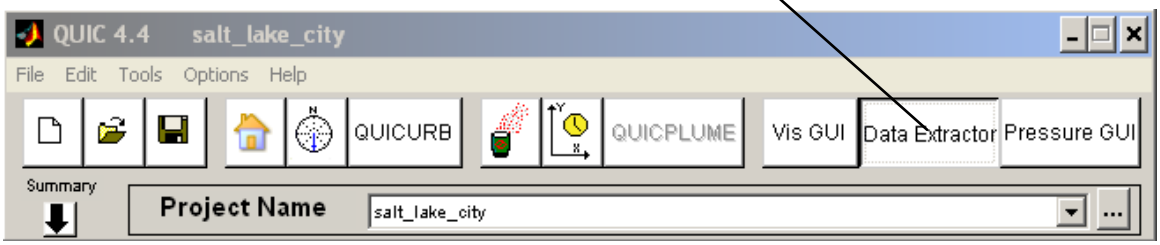

Sequences can be copied, deleted, and reordered to produce the desired movie. After manipulating the sequences, select the desired sequences and press the 'Preview Movie' button. This will replay the selected frame sequences in a new figure. After the movie has been previewed the 'Finish Movie' button at the bottom of the GUI becomes active. Pressing this button will prompt the user to save the animation in the selected file format.

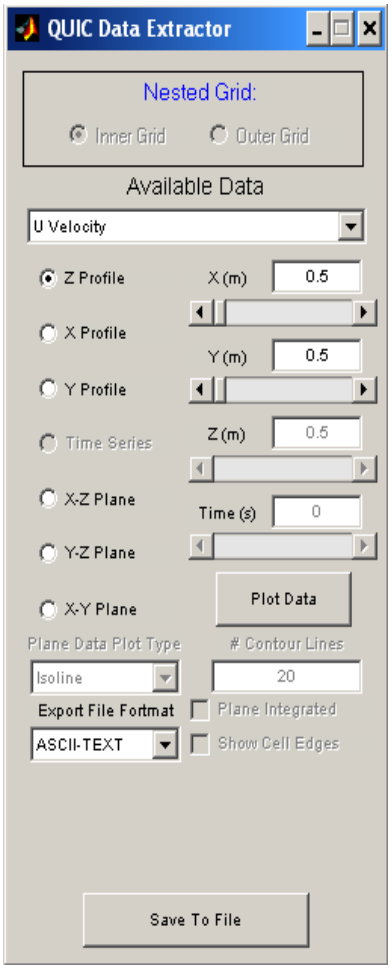
# DATA EXTRACTOR

# QUIC Data Extractor

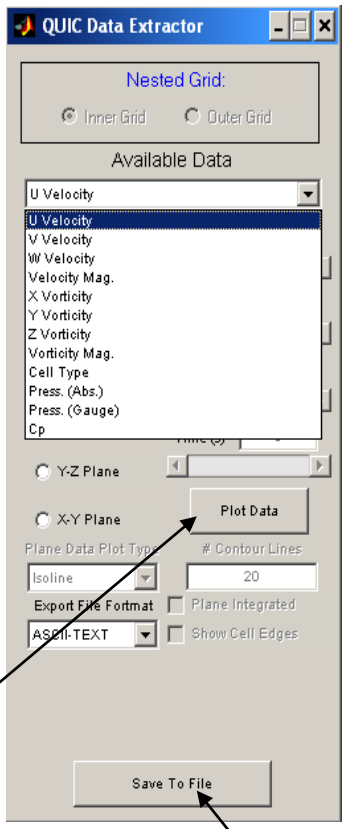
The user has the option to output concentration data in column format. Vertical, horizontal, and time series profiles can be saved, in addition to a plane of data. The text output option is useful when wanting to compare model results to experimental data, for example.

- In the main QUIC window select 'Data Extractor'
- The QUIC Data Extractor Window opens.



- The Data Extractor window has the 'Available Data' option. Using the 'Available Data' pull down menu shows the data that can be plotted and saved.




- After selecting certain flow parameter, it can either be plotted using the 'Plot Data' option or it can be saved using the 'Save to File' option.

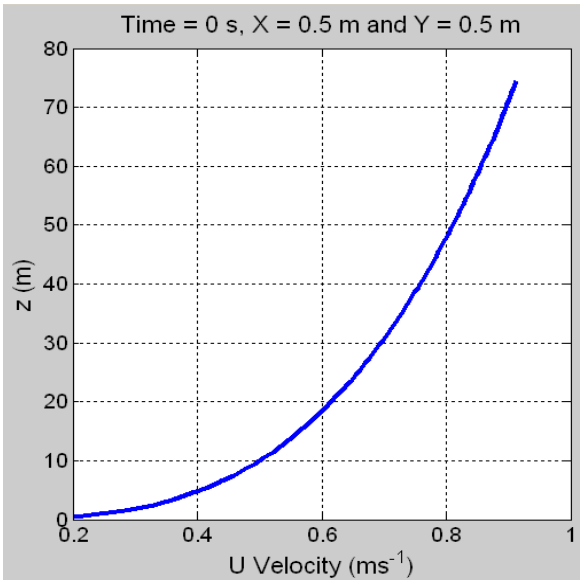
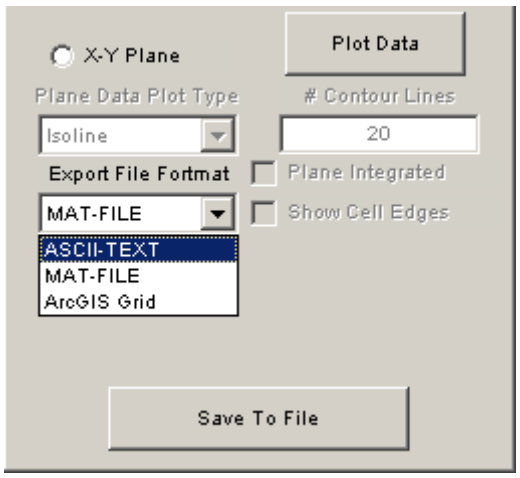
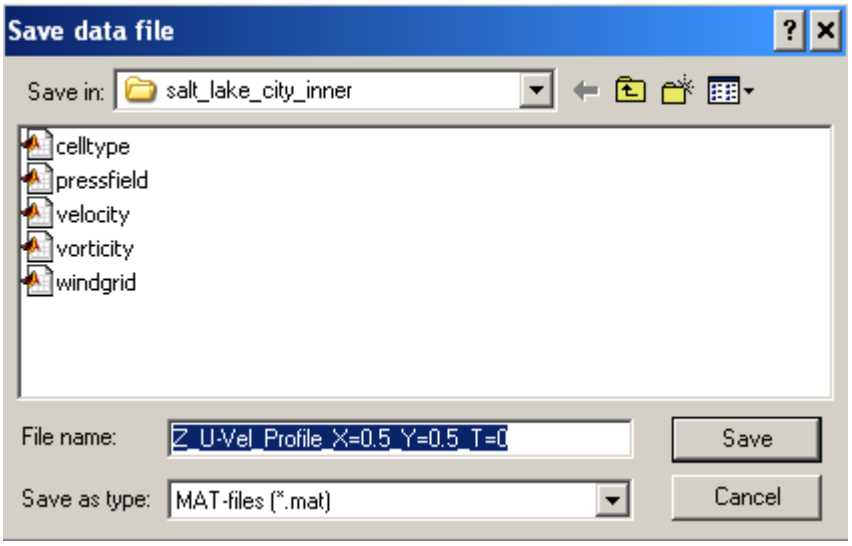



Figure shows the 'U' velocity plotted using the 'Plot Data' option

- The data can be saved in three formats. They are:
    - ASCII
    - MAT files
    - ArcGIS




- Select any format and press the 'Save to File' button
- On pressing the 'Save to File' button, the user will be a directed to folder where he/she would like to save the files

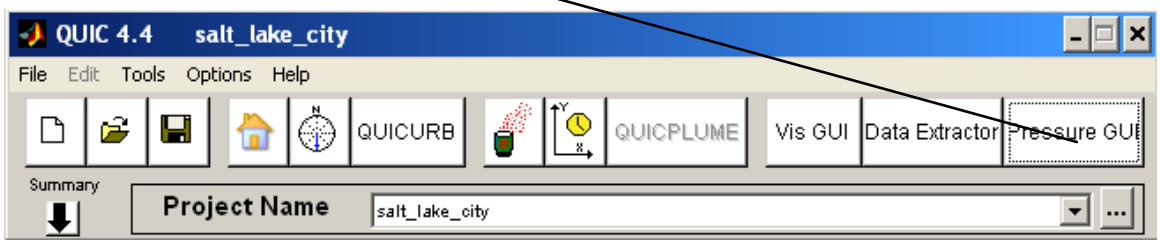


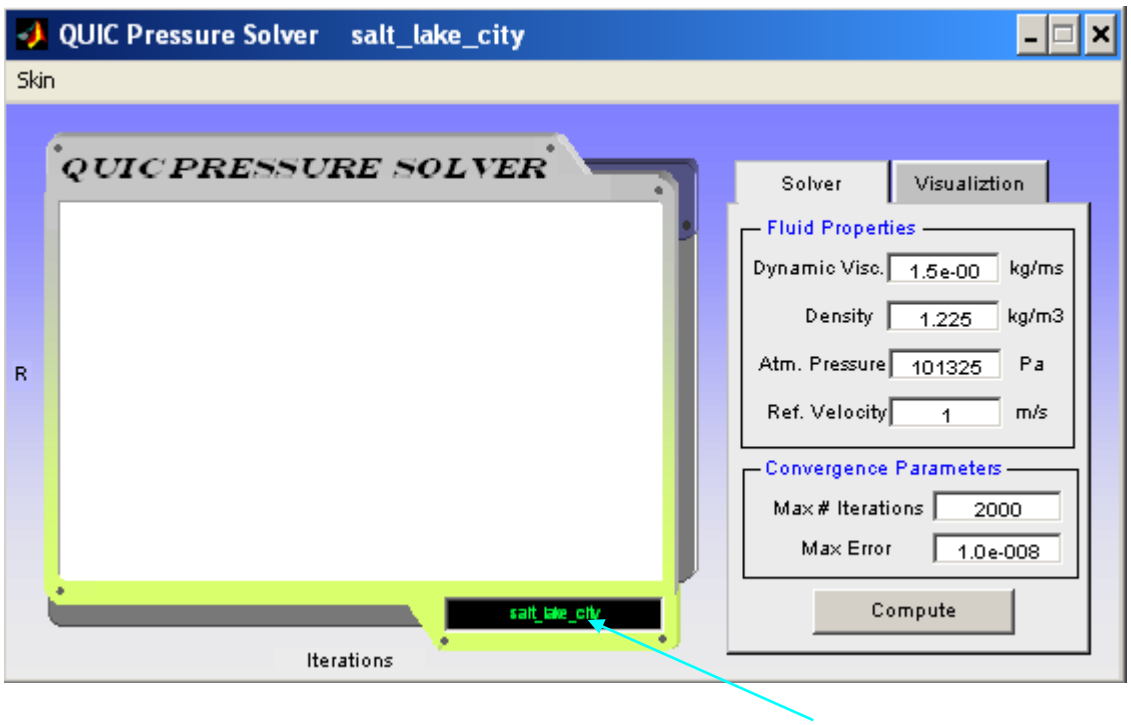

Make selections for desired data output and press "Save" button. The popup window for naming the file opens. The default filename contains descriptive information on the profile or planar coordinates.

# PRESSURE GUI

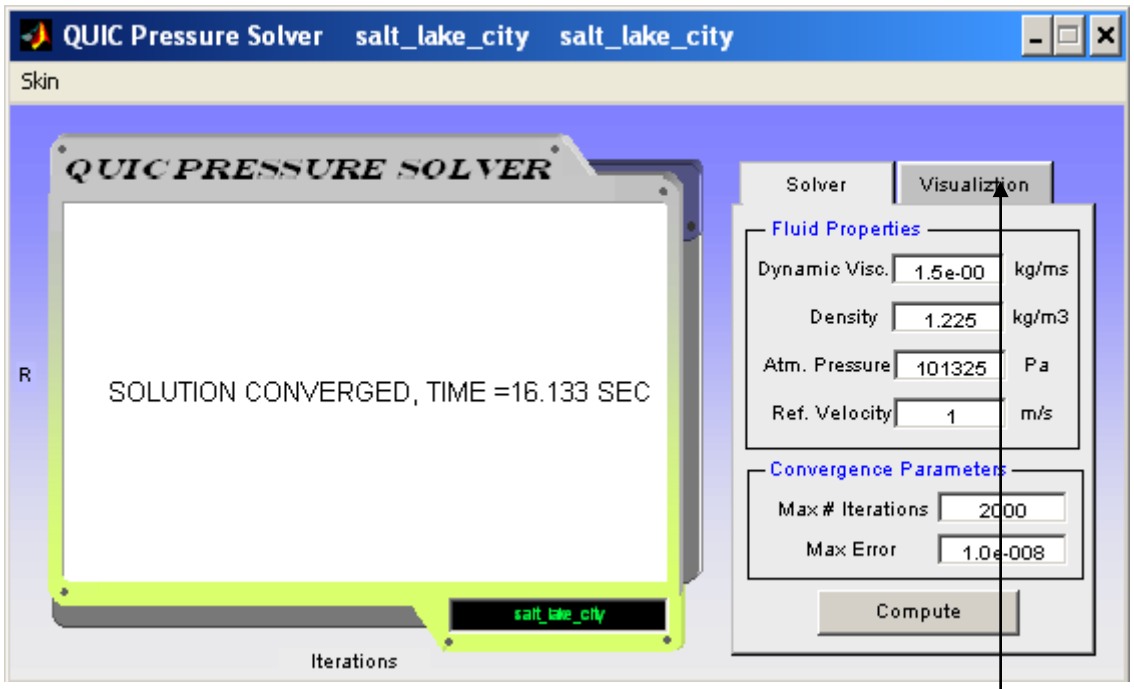
# QUIC Pressure Solver

- Selecting 'Pressure GUI' from the main QUIC window launches the QUIC Pressure Solver

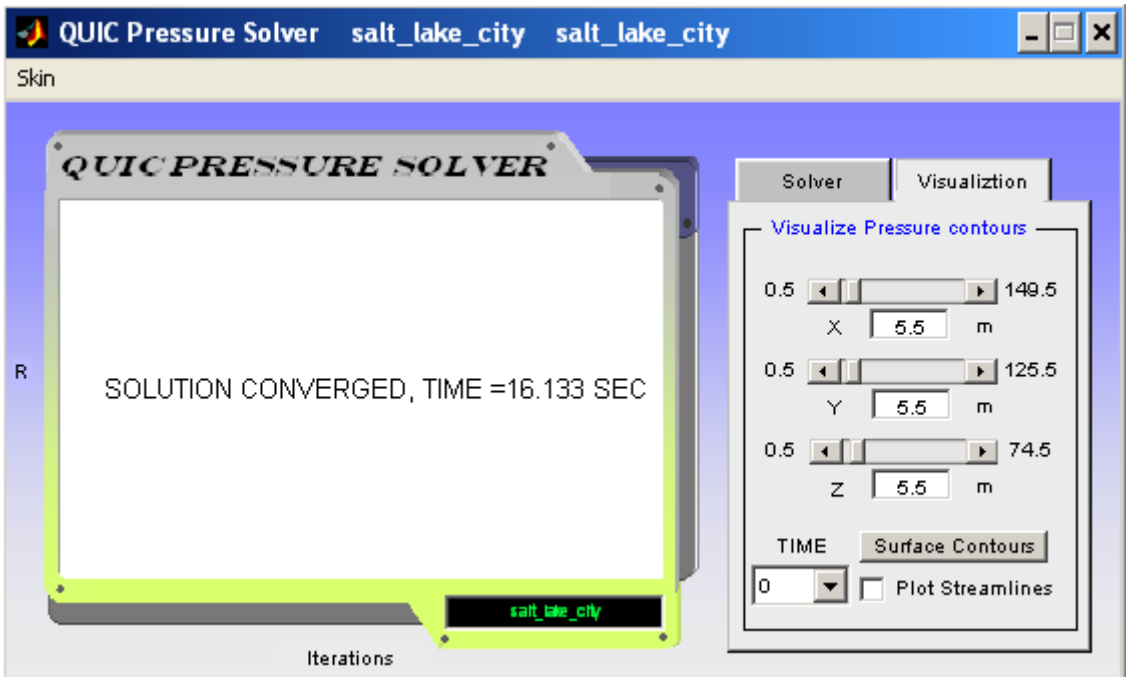



- The name of the project can be seen in window at the bottom.
- Various fluid properties such as the dynamic viscosity and density can be specified in the window to the right.
- The user can specify the maximum number of iterations over which the Pressure Poisson Equation should be solved. The default value is 500.
- The user can also specify the threshold for maximum error. The default maximum error is 1e-6.
- After setting all the parameters press the 'Compute' button at bottom of the screen
- The QUIC Pressure Solver solves the Pressure Poisson Equation for the given domain and computes the pressure field.




- To visualize the pressure field for the given domain, choose the 'Visualization' option.



- To view the pressure contours along a particular plane, enter the value of the plane (X, Y, or X) or scroll with the sliders until the pressure field along the desired plane can be seen.

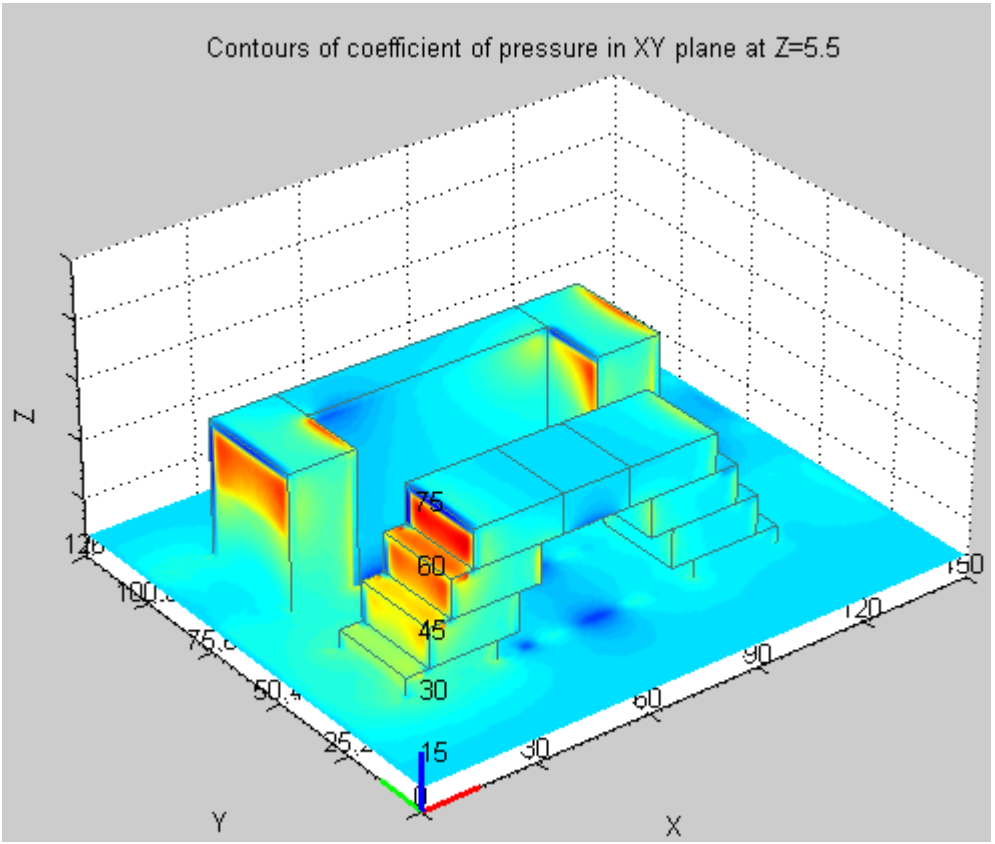
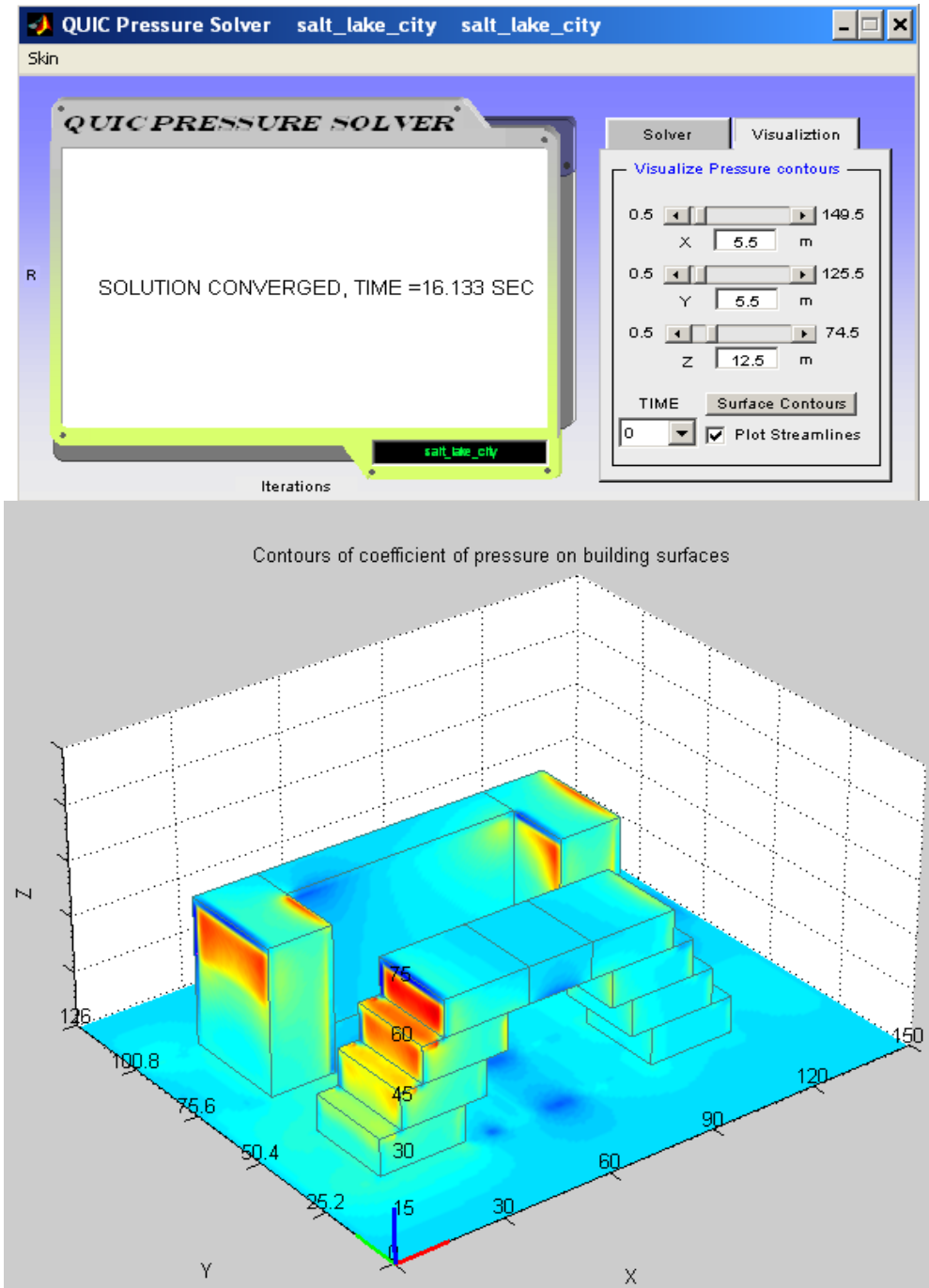
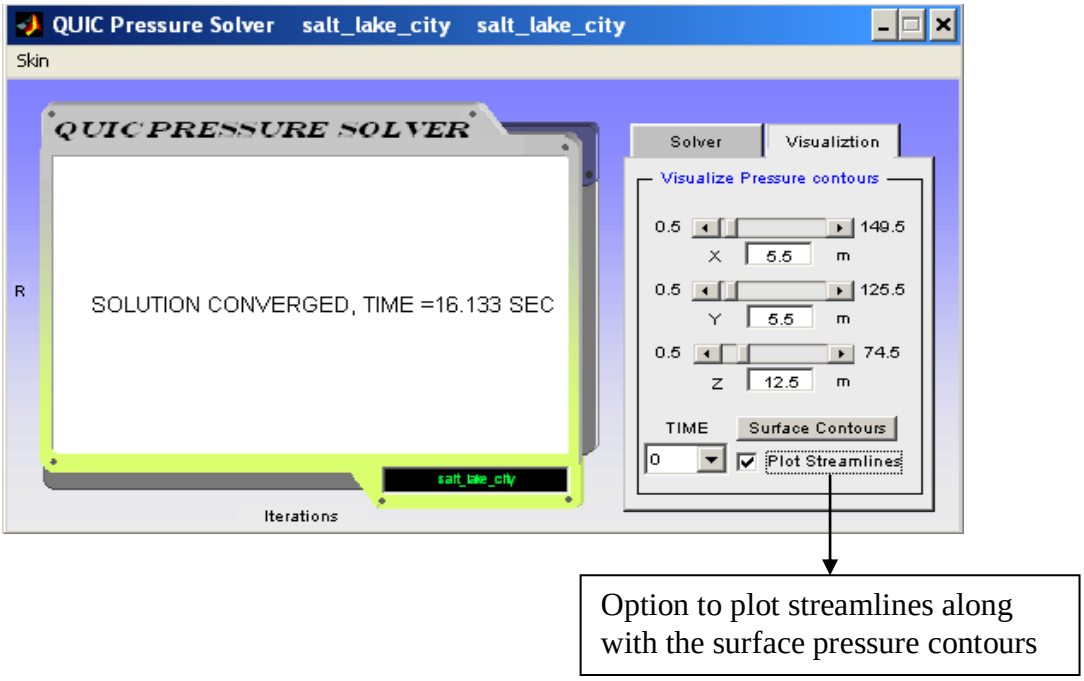



Figure shows the pressure field in the 'XY' plane

- To view the pressure field on the building surfaces, choose 'Surface Contours'




- To view the streamlines along with the surface pressure contour, select the 'Plot streamlines' option



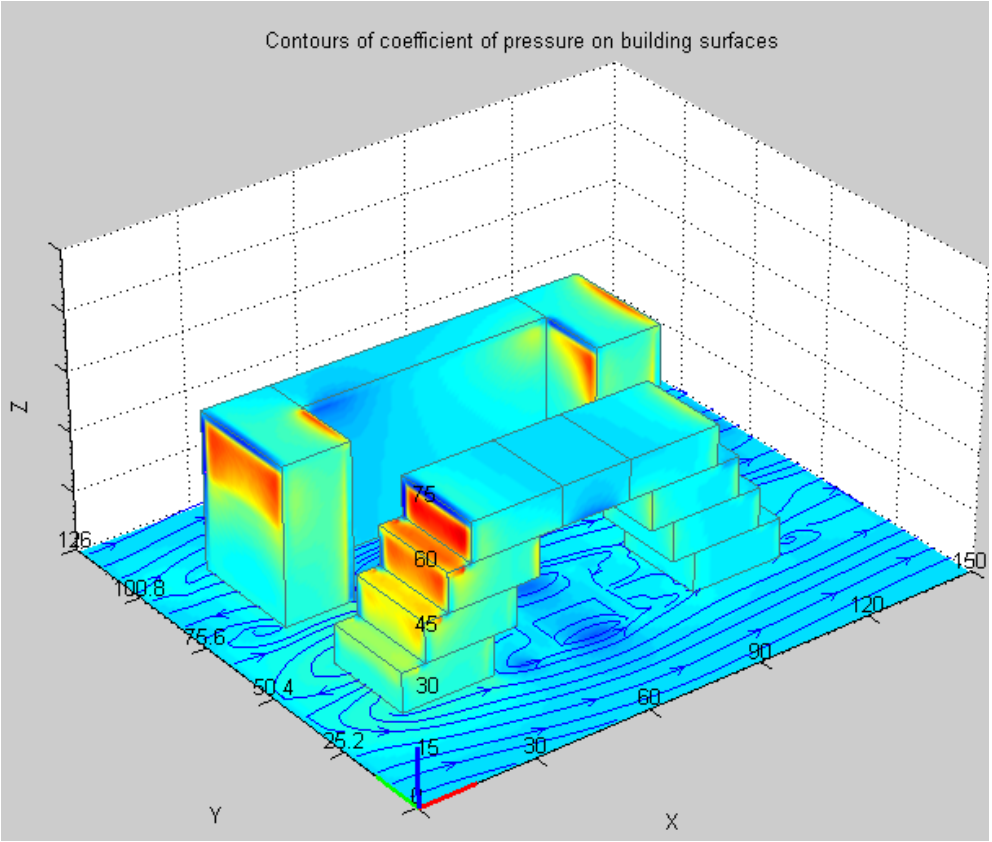
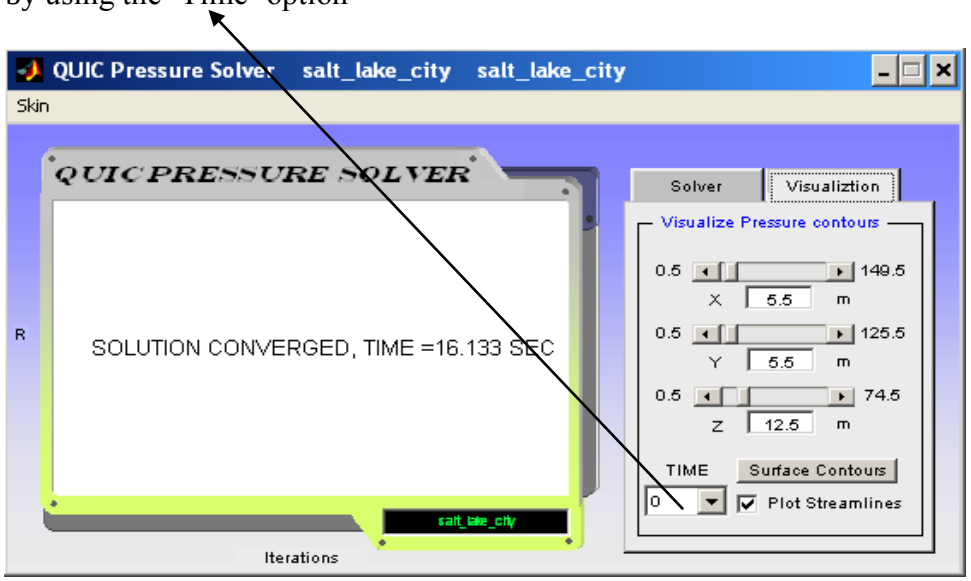




Figure shows streamlines plotted along with the surface pressure contour

- In case multiple time averaged velocity profiles were input through the MET Generator, the pressure fields for the different inflow velocity fields can be seen by using the 'Time' option




## Example Projects

Within the Projects folder are two projects: one created during the creation of the QUIC Start Guide and the other from a building database. A project for a three block by three block area of downtown Salt Lake City and a project for a stacked building are included. Once loaded they can be visualized and edited.

## References


Bagal, N., E. Pardyjak, and M. Brown, 2003: Improved upwind cavity parameterization for a fast response urban wind model, AMS Conf. on Urban Zone, Seattle, WA, 3 pp.

Booth, T., 2003: Testing and implementation of urban and plant canopy parameterizations into the Quick Urban & Industrial Complex (QUIC) dispersion modeling system, Univ. of Utah Technical Report, 8 pp.

Bowker, G., S. Perry, and D. Heist, 2004: A comparison of airflow patterns from the QUIC model and an atmospheric wind tunnel for a two-dimensional building array and multi-block region near the World Trade Center site, 5th AMS Urban Env. Conf., Vancouver, B.C., 6 pp.

Clark, J. and P. Klein, 2006: Implementation of a traffic-produced turbulence scheme into the fast-response model QUIC, 6th AMS Urb. Env. Symp., Atlanta, GA, 7 pp.

Gowardhan, A., M. Brown, M. Williams, E. Pardyjak, 2006: Evaluation of the QUIC Urban Dispersion Model using the Salt Lake City URBAN 2000 Tracer Experiment Data- IOP 10. 6th AMS Symp. Urban Env., Atlanta, GA, LA-UR-05-9017, 13 pp.

Gowardhan, A., Gowardhan, A., M. Brown, M. Nelson, E. Pardyjak, D. DeCroix, D., 2006: Evaluation of the QUIC Pressure Solver using wind-tunnel data from single and multi-building experiments. 6th AMS Symp. Urban Env., Atlanta, GA, LA-UR-05-9016, 12 pp.

Pardyjak, E. and M. Brown, 2002: Fast response modeling of a two building urban street canyon, 4th AMS Symp. Urban Env., Norfolk, VA, May 20-24 2002, LA-UR-02-1217.

Pol S., N. Bagal, B. Singh, M. Brown, and E. Pardyjak, 2006: Implementation of a rooftop recirculation parameterization into the QUIC fast response wind model, 6th AMS Urb. Env. Symp., Atlanta, GA, LA-UR-05-8631, 19 pp.

Singh, B., E. Pardyjak, and M. Brown, 2006: Testing of a Far-wake Parameterization for a Fast Response Urban Wind Model, 6th AMS Urban Env. Conf., Atlanta, GA, LA-UR-05-###, 11 pp.

Williams, M., M. Brown, B. Singh, & D. Boswell, 2004: QUIC-PLUME Theory Guide, LA-UR-04-0561, 22 pp.

Williams, M., M. Brown, D. Boswell, B. Singh, and E. Pardyjak, 2004: Testing of the QUIC-PLUME model with wind-tunnel measurements for a high-rise building, 5th AMS Urban Env. Conf., Vancouver, B.C., LA-UR-04-4296, 10 pp.

# Appendix A

Files contained in QUIC V 4.5 code folder.

AlignView.p
DataImportDefaults.dat
axistoggle.bmp
axistoggledwn.bmp
axistoggleup.bmp
back_plain.bmp
backgrd.bmp
balloon.bmp
checkinput.p
citybuilder.fig
citybuilder.p
cityoptions.fig
cityoptions.p
cityplot.p
cityplot_params.p
colorset_generator.p
convert_QPR_pressfield.p
convert_QP_condos.p
convert_QP_confield.p
convert_QP_depct.p
convert_QP_partfield.p
convert_QP_turbfield.p
convert_QP_turbtype.p
convert_QU_velocity.p
convert_QU_velocity_ascii.p
convert_QU_vorticity.p
copyright.txt
correlate.fig
correlate.p
create_quicgui_viewmenus.p
dataplot.bmp
default_QP_output_files.inp
default_options.info
default_params.p
dimension.fig
dimension.p
domain_extend.fig
domain_extend.p
dplane.bmp
handpoint.bmp
infilt.bmp
infiltration_gui.fig
infiltration_gui.p
interpstreamspeed2.p
isencompassing.p
isone.p
material_library
nonrmcurl.p
optionapply.p
particlemeshgridmapping.p
pent.bmp
pic3d.p
plot_2d_profiles2.p
plot_2d_qp_profile.p
plot_3dcomp_profiles2.p
plot_3ddir_profiles2.p
plot_all_streamlines2.p
plot_con_contour_isolines.p
plot_con_contours.p
plot_cones2.p
plot_contour_isolines.p
plot_contours2.p
plot_current_point2.p
plot_lct_contour.p
plot_particles.p
plot_pathlines2.p
plot_plume.p
plot_qp_profile.p
plot_streamlines2.p
plot_surface_deposition.p
plot_turbulence.p
plot_vectors2.p
plot_vorticity2.p
plume_phys.fig
plume_phys.p
positionfixer.p
pressure_viz.p
qp_contourslice.p
quic.p
quic_audio.p
quic_cbar.p
quic_coneplot.p
quic_data_extractor.fig
quic_data_extractor.p

quic_help.fig
quic_help.p
quic_imread.p
quic_imwrite.p
quic_met.fig
quic_met.p
quic_movie.fig
quic_movie.p
quic_pressure.fig
quic_pressure.p
quic_quiver3.p
quic_sensor.fig
quic_sensor.p
quic_source.fig
quic_source.p
quic_stuff.html
quic_stuff_contents.html
quic_stuff_details.html
quic_stuff_summary.html
quic_tetraplot.p
quic_vis.fig
quic_vis.p
quic_wind_importer.fig
quic_wind_importer.p
quicfigcameratoolbar.p
quicgui.fig
quicgui.p
quicplume_LINUX_32.exe
quicplume_LINUX_64.exe
quicplume_MAC.exe
quicplume_WIN.exe
quicplume_sim_setup.fig
quicplume_sim_setup.p
quicpressure_LINUX_32.exe
quicpressure_LINUX_64.exe
quicpressure_MAC.exe
quicpressure_WIN.exe
quicurb_LINUX_32.exe
quicurb_LINUX_64.exe
quicurb_MAC.exe
quicurb_WIN.exe
read_QPR_params.p
read_QP_buoy.p
read_QP_erad.p
read_QP_fileoptions.p
read_QP_indoor.p
read_QP_materials.p
read_QP_params.p
read_QP_particlesize.p
read_QP_source.p
read_QU_buildings.p
read_QU_fileoptions.p
read_QU_metparams.p
read_QU_simparams.p
read_info.p
read_input_dat.p
read_materials_lib.p
read_project_file.p
reduce_space.p
rotate_object_data.p
size_pic.p
slice_quic.p
str2cmap.p
urb_phys.fig
urb_phys.p
vol3d.p
weather_vain.bmp
weather_vane.bmp
write_QPR_params.p
write_QP_buoy.p
write_QP_erad.p
write_QP_fileoptions.p
write_QP_indoor.p
write_QP_materials.p
write_QP_params.p
write_QP_particlesize.p
write_QP_source.p
write_QU_buildings.p
write_QU_fileoptions.p
write_QU_metparams.p
write_QU_simparams.p
write_info.p
write_project_file.p
write_slice_vel_field.p
xyplane.bmp
xyuvcheck.p
xyzuvwcheck.p
xyzvcheck.p
xzplane.bmp
yzplane.bmp
zwake.p