Table of Contents

Tutorials

This is the summary of the tutorials.

Tutorial list:

- Fluidized Bed
- Silo Flow

Title: Analysis of dynamic systems: Fluidized bed

Author: Prashant Gupta

Date: February 28, 2014

Introduction

This document describes pre and post processing of fluidized bed system using P4 tool developed in University of Edinburgh. Brief description of 3D DEM-CFD simulation is presented in following section, followed by pre and post processing of particle data (DEM). This case study was done as a post processing of validation data. High quality validation experimental data obtained was used for comparisons. Time averaged particle velocity and solid fraction profiles/contours were obtained. This case study would demonstrate how to post process DEM data obtained from simulations to compare against experiments. Millimeter size particles are subjected to a fluidizing velocity much greater than minimum fluidization velocity. This leads to transient bubbling/slugging regime. Bubbles are formed around the distributor plate, rise and coalesce and collapse at the free surface of the bed. Hence, bubbling and mixing behavior leads to lower solid fraction in the middle of the bed and lower particle velocities at the walls. Typical snapshot of such a phenomenon can be seen in figure 1.

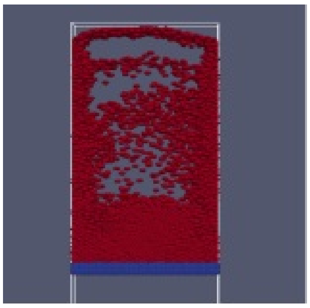


Figure 1: Typical snapshot of DEM-CFD simulation of gas-solid fluidized bed with particle size 1.2 mm and density 1000 kg/m at inlet velocity 0.9 m/s

Input: DEM data in this tutorial is extracted from Lammps using dump class available in Lammps-ed version. For more details, refer to the P4 Manual on how to dump data from lammps-ed.

Output: This tutorial is aimed at explaining meshing required and outputting useful results such as smooth contour plots and line plots for dynamic granular system such as fluidized bed. After this tutorial, Following output would be obtained:

- Smooth contour plots of solid fraction and solid velocities.
- Line plot of these quantities at different heights (as measured in experiments).

3-D DEM-CFD simulation of a bubbling fluidized bed was done by coupling Lammps and OpenFOAM outputting useful particle data (DEM) at high sampling frequency. Table 1 gives the simulation parameters including particle size, density and domain size.

Pre-processing

Domain of the problem is a pseudo 3-D cuboid which is created according to the simulation parameters given in the table. Idea is to create a rectangular section with x from 0.0 to 0.044 and y from 0.0 to 0.12 and extrude in z direction. This can be done by following these step:

- Create a rectangle by `Menu ⇒ Geometry ⇒ Object ⇒ Rectangle` and enter first centre point as (0.0 0.0 0.0), ENTER, second point as (0.044 0.12 0.0). Always useful to use zoom frame for a better visibility and fit to the screen.
- Extrude this in Z direction by `Menu ⇒ Utilities ⇒ Copy`. From the toolbox select: Entities Type: Surfaces Transformation:

Translation First point (0.0 0.0 0.0) and second point (0.0 0.0 0.01). Do extrude Volumes Select. (This will lead to a geometry like figure 2)

- Next step is to mesh the geometry: Regular structured mesh with hexahedral elements will be used in the present example.

Select `Menu ⇒ Mesh ⇒ Structured ⇒ Volume ⇒ Assign number of cells` and then select the geometry and press esc. Enter number of cells to assign to lines as : 1 and select any line on Z axis, press Esc. proceed with assigning number of cells as 11 in X axis and 40 on Y axis. This defines the spatial scale at which the coarse grained quantities would be calculated. To assign elements as hexahedral : Again select `Menu ⇒ Mesh ⇒ Element type ⇒ Hexahedra` and select the volume, press Esc.

`Menu ⇒ Mesh ⇒ Generate Mesh` (or `Ctrl-g`), press `OK` & `View Mesh`. This should look like figure 3. Change view to X-Y. Save the project in `Menu ⇒ File ⇒ Save`.

Particle Info	**Value**
Number of Particles	9240
Diameter (mm)	1.2
Sphericity	1
Particle Density (kg/m3)	1000
Geometry	**Value**
Bed width (m)	0.044
Bed height (m)	0.12
Bed thickness (m)	0.01

Table 1: Particle information

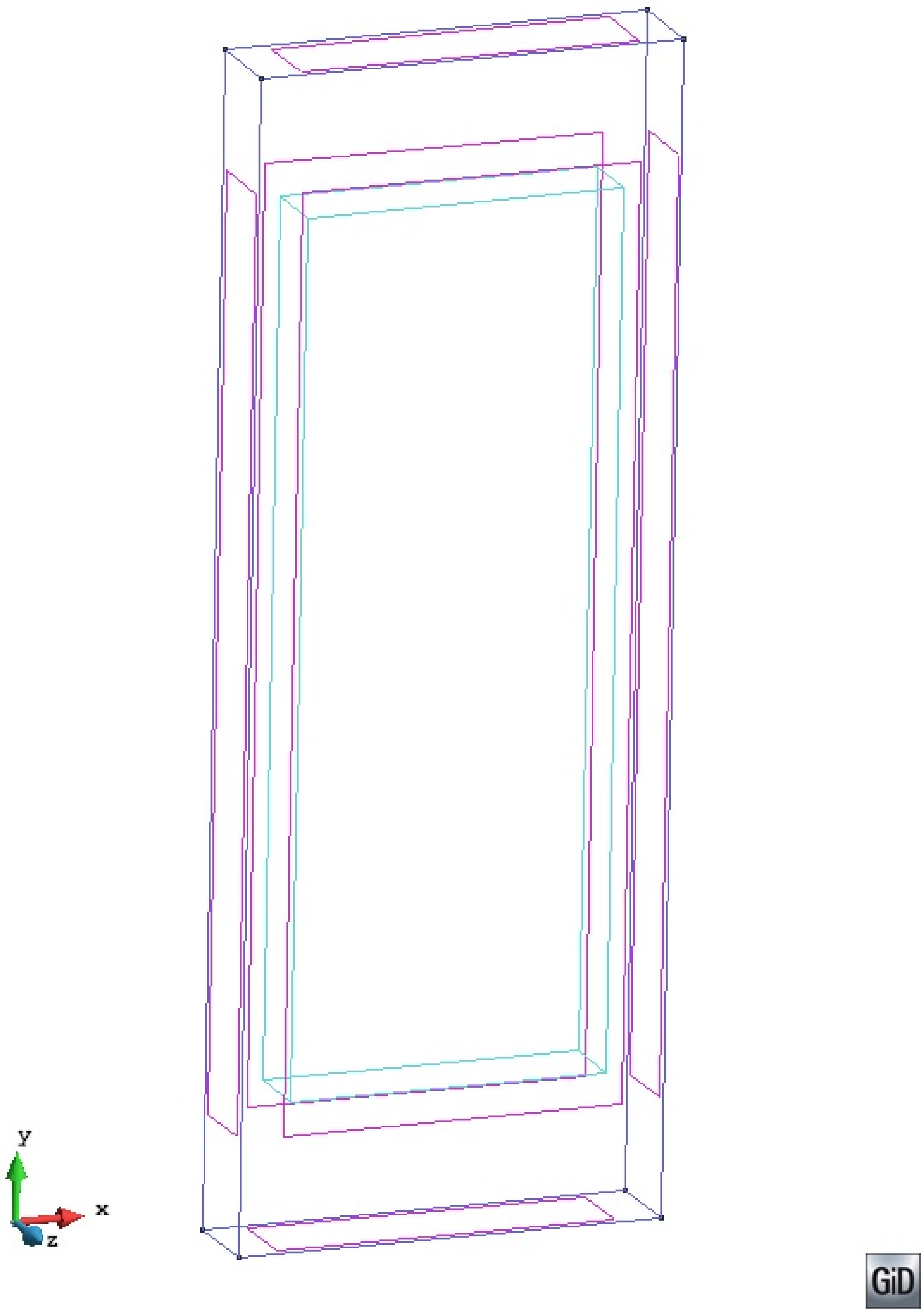


Figure 2: Domain for meshing as produced by GiD

P4 settings

Dump files are created by lammps as dump.p3p, dump.p3c and dump.p3w as particle definition file, contact data and particle-wall contact data respectively. Next P4 toolbox is loaded by going to `Data ⇒ Problem type ⇒ p4-v0.4` (whichever is the latest version). This will load P4 toolbox. Click on the CG preferences button just below P4 symbol. Following settings are used in the P4 tool box for present example.

Input

- `FILENAME` : Browse to the destination of dump.p3p and select.
- `READ CONTACTS` : NO (Not for the present case)
- `READING TIME STEPS` : ALL
- `STEP FREQUENCY` : 1
- `FILTERING ID` : Filter Group ID 3 (these are fixed particles in the simulations for present case)

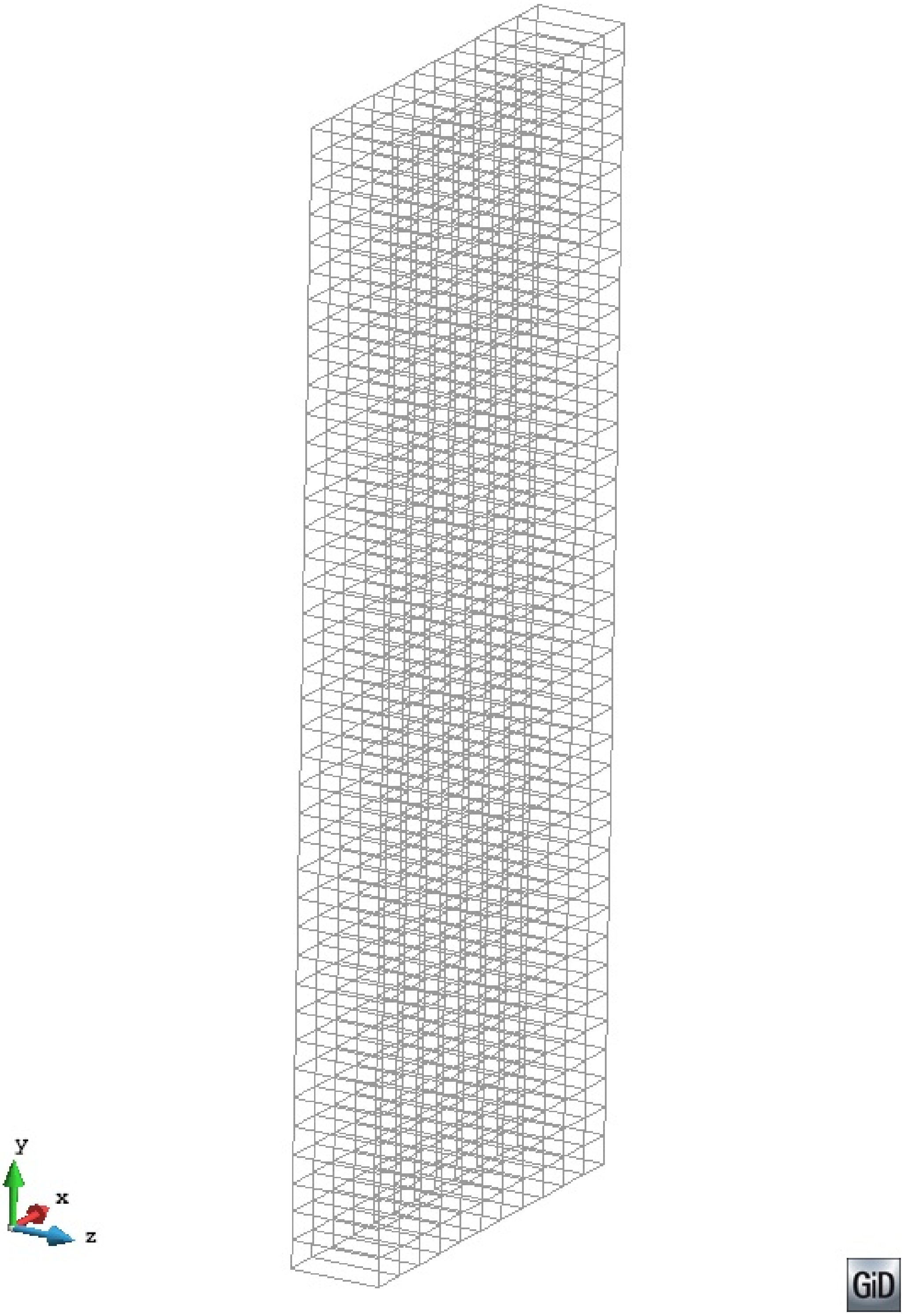


Figure 3: Regular structured mesh for post processing

Time Averaging

- `USE TIME AVERAGE` : Yes
- `TIME AVERAGE` : ALL

Spatial Averaging

- `TYPE` : Binning

Output

- `FILE TYPE` : GID BINARY is chosen in this particular example.
- `PRINT PARTICLES` : YES option is checked to visualise particles.

After selecting appropriate settings, press `SELECT` and save the project file once more. Start process, after finished click on post process. Load the post-process file (.post.bin) from the post processing tool box. Data is now loaded to be visualized. This example primarily brings out time and spatial averaged contour plots of a bubbling fluidized bed.

Post Processing

These can be visualized by: Go to `Menu > View Results > Smooth Contour Fill > Solid Fraction or Velocity X,Y` . This would provide us time averaged smooth contour plots like in figure 4.

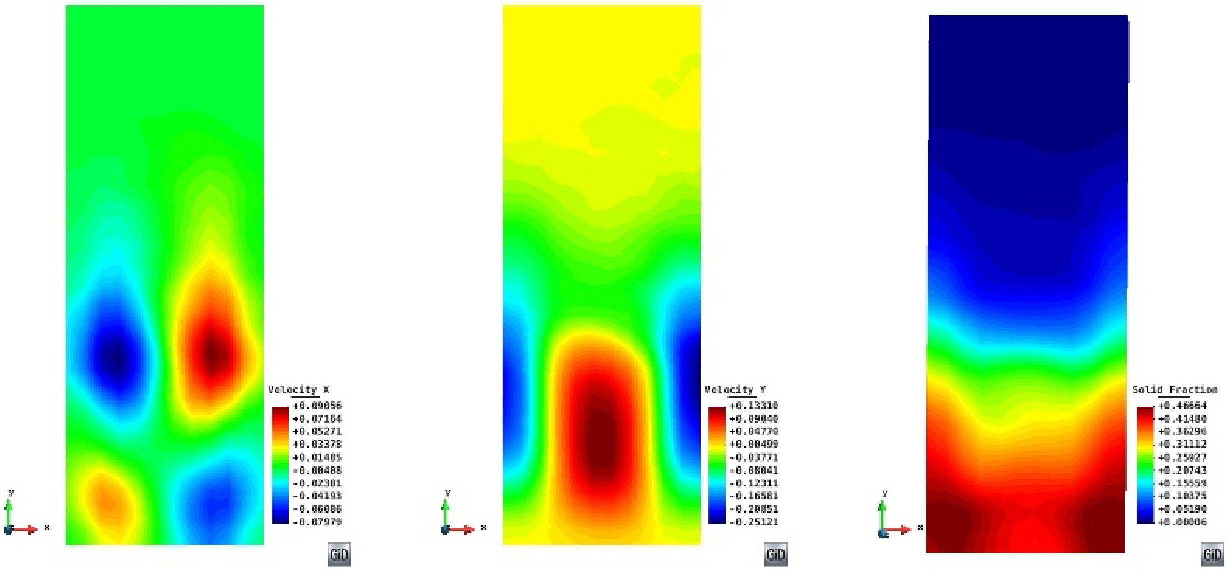


Figure 4: Time averaged contour plots of bubbling fluidized beds showing (a) Averaged velocity in axial direction (b) Averaged vertical velocity (c) Averaged solid fraction

For obtaining graph (say a line plot) 5, Use GiD utilities to make line plots at different heights Y = 0.0164 m and 0.0312 m. Go to: `Several graphs option > Line graph > Solid fraction` . This will prompt to select points, enter coordinates as *(0.0, 0.0164)* and *(0.044, 0.0164)* to plot line variation graph at height 0.0164 m. (Refer to GiD manual if more information is required)

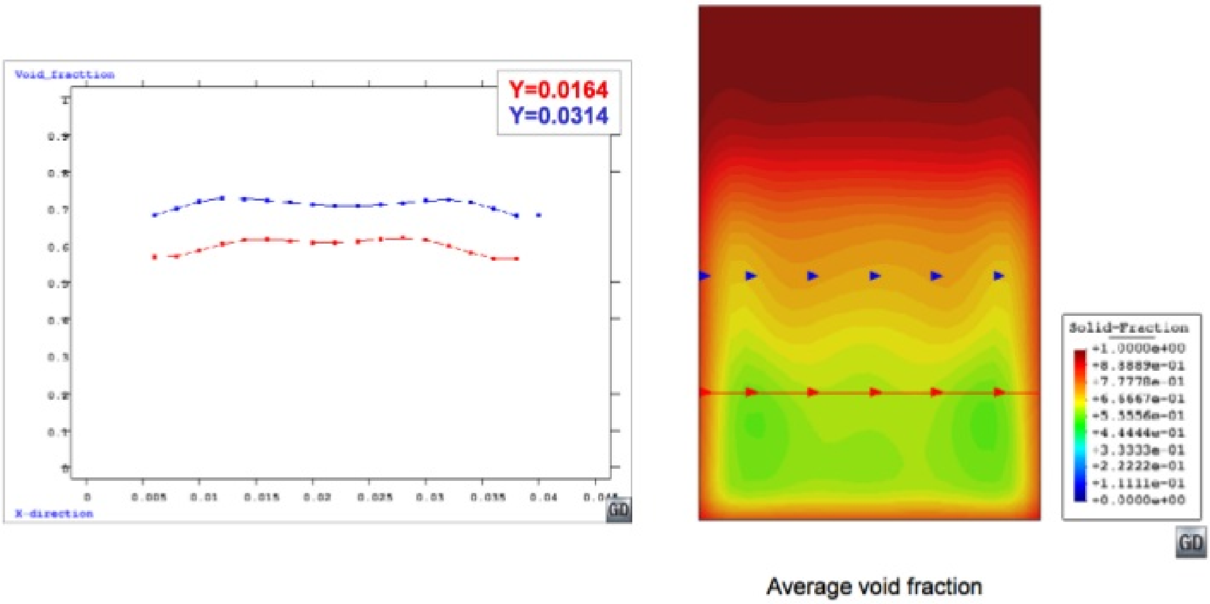


Figure 5: Line plot variation in axial direction for solid fraction at heights 0.0164 and 0.0312 m

Title: Silo flow

Author: Pratap Kasina

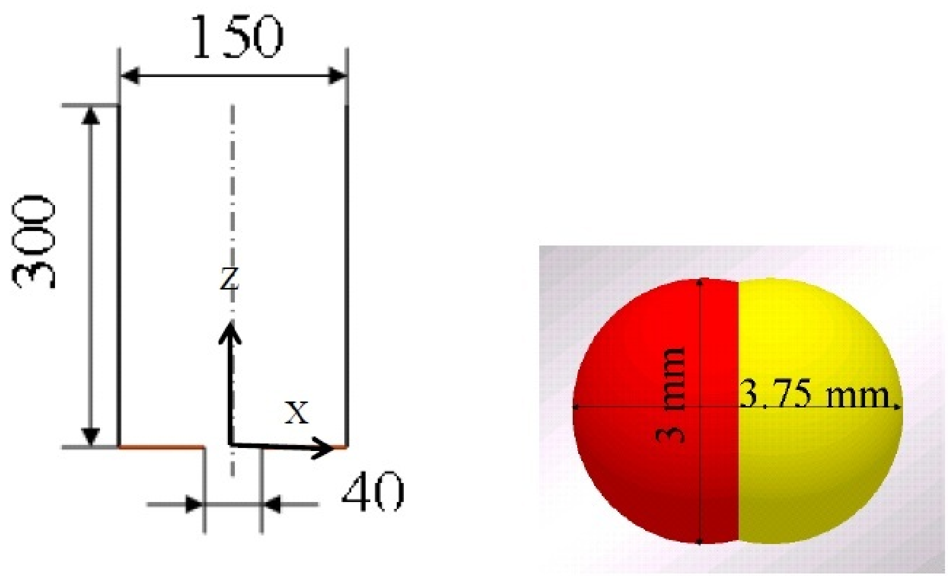
Date: February 2014

Introduction

The objective of this tutorial is to demonstrate how to convert the particle scale data that is generated from DEM to a scale at which the experimental data is usually collected. Particle scale data is averaged spatially using coarse graining technique and the theoretical background of this technique is discussed in P4 manual. The flow patterns of granular solids in flat bottom silo is analysed using P4 in order to validate the model by comparing the results with experimental observations.

The discharge of non-spherical particles in flat bottom silo was simulated using EDEM 2.5.1 software. The silo is of 40 mm thick with an outlet width of 40 mm as shown in Fig 1a. The shape of the particle was modelled as combination of two spheres of diameter 3 mm clumped together to give an aspect ratio of 1.25 as shown in Fig.1 (b). The silo is centrally filled from the top and the filling is considered to be completed when the total kinetic energy of the system reaches below 1E-08 J which is negligible when compared to the total K.E of the system during discharge (J). The material is them allowed to discharge by removing the outlet plate.

The data is exported at 200 Hz from EDEM during 20-30 % of material discharged from the silo. The results which are time averaged during 20-30 % of material discharged from the silo are reported here in the form of contour and graphical plots. The following sections describe the procedure to achieve the above mentioned objective with the use of GiD software combined with P4 plugin which includes three main steps: pre- processing, averaging the data, post processing.



(a) (b)

Fig.1 (a) Schematic diagram of flat bottom silo (b) Particle shape

Figure 1: (a) Schematic diagram of flat bottom silo (b) Particle shape

Pre-processing

The pre-processing includes creating the geometry of the system followed by discretisation of the domain with one of the various methods available in GiD.

Create a box

- `Geometry ⇒ Create ⇒ Object ⇒ Rectangle`.
- Specify the corner points of a rectangle: First corner point is (-0.075, 0.04,0) and Second corner point is (0.075,0,0)
- Volume can be created by sweeping the rectangle along Z axis. Select `Utilities ⇒ Copy`. In the Copy window, select Surfaces and Translation from the drop down lists next to Entity types and Transformation. Select volumes from the list next to Do Extrude
- The height of extrusion is specified by first point (0, 0, 0) and second point (0, 0, 0.3).
- Click Select button and the click the rectangle in the viewer window and press Escape.
- `Select Zoom ⇒ View ⇒ Frame` to fit the box in the view window. This can also be accessed directly from the standard tool bar as shown in Fig 2.

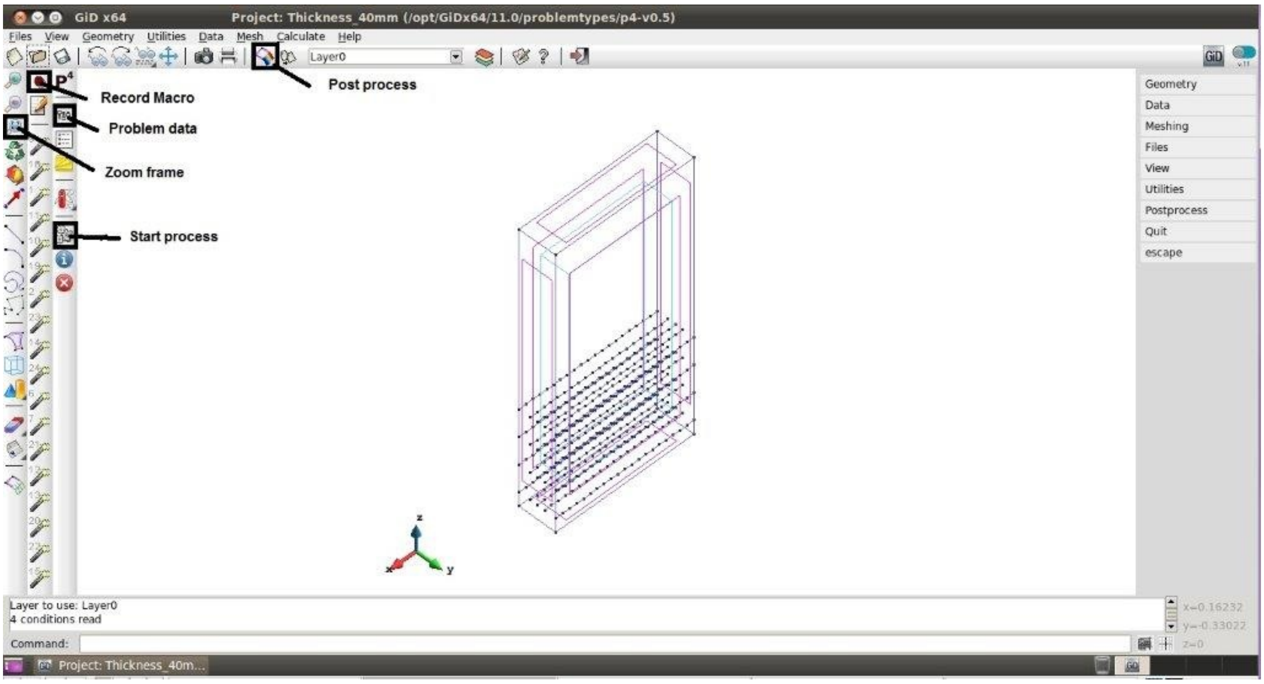


Figure 2: Layout of GiD pre-processor

Create lines for graph

It is recommended to create additional lines in the domain along which the results are expected to plot. This avoids any interpolation of data by GiD while plotting a line graph (refer to section 4.5) if there are no nodes on the line. The P4 then calculates the information on the nodes of these lines.

Having decided the location of lines for graphs it is easy to generate all of them at once by recording a macro while create one line and then edit the same macro to create multiple lines in one click. The process in described below:

- Click on record macro button (as shown in Fig 2)
- `Geometry ⇒ Create ⇒ Straight line`. Input the coordinates of the line from command window as (-0.075, 0.015, 0) and (0.075, 0.015, 0).
- `Geometry ⇒ Edit ⇒ Divide ⇒ lines ⇒ Number of divisions`. Enter 20 and click ok. This prompts the user to select lines from the viewer window. Select the line created in step 2 and press escape (In GiD, pressing Escape key ends a command or finishes input definition for any selected function).
- Stop recording the Marco by click the same button. This saves a macro file (displayed next to the record macro button) and right clicks the file and select edit widow: copy the first command of the macro multiple times and change the coordinates of the lines in commands. Create four equally spaced lines along Y direction at heights of 0.015, 0.045, 0.075, and 0.105 as shown in Fig 3.

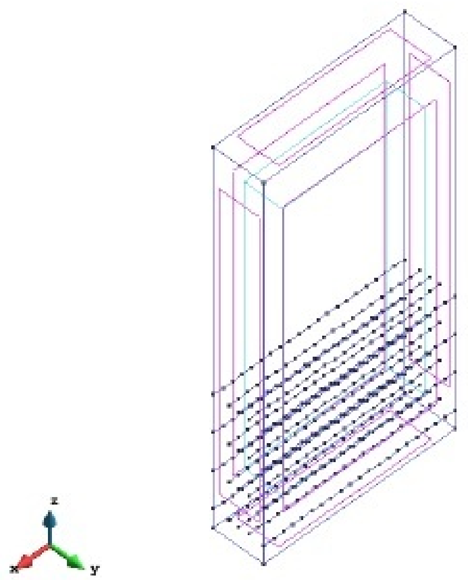


Figure 3: Problem domain showing the lines for graphs

Meshing

The volume is meshed with structured tetrahedral elements. The user has also have options to mesh the volumes with other element types including hexahedral and prisms. The following procedure demonstrates how to generate structured mesh with tetrahedral elements.

Define the mesh size

Select `Mesh ⇒ Structured ⇒ Volumes ⇒ Assign number of cells`.

Select the volume by clicking on Cyan coloured boundary (By default, volumes are represented by this colour) with mouse and pressing Escape pops ups Enter value window. Specify the number of cells to be assigned along each axis. Enter 20 and click assign and then select any line parallel to X-axis which then automatically selects the rest of lines parallel to X-axis and press Escape. Repeat the same procedure and assign 20 and 4 number of cells along Z and Y axis respectively. Close the enter the value window after the finishing the assignment.

Define the element type

Select `Mesh ⇒ Element Type ⇒ Tetrahedral`. This prompts you to select the volumes to assign the tetrahedral elements. Select the volume. Press Escape

Generate mesh

Select `Mesh ⇒ Generate mesh`. In the mesh generation window tick the get parameters from the model option and then select OK. This generates the mesh as shown Fig 4.

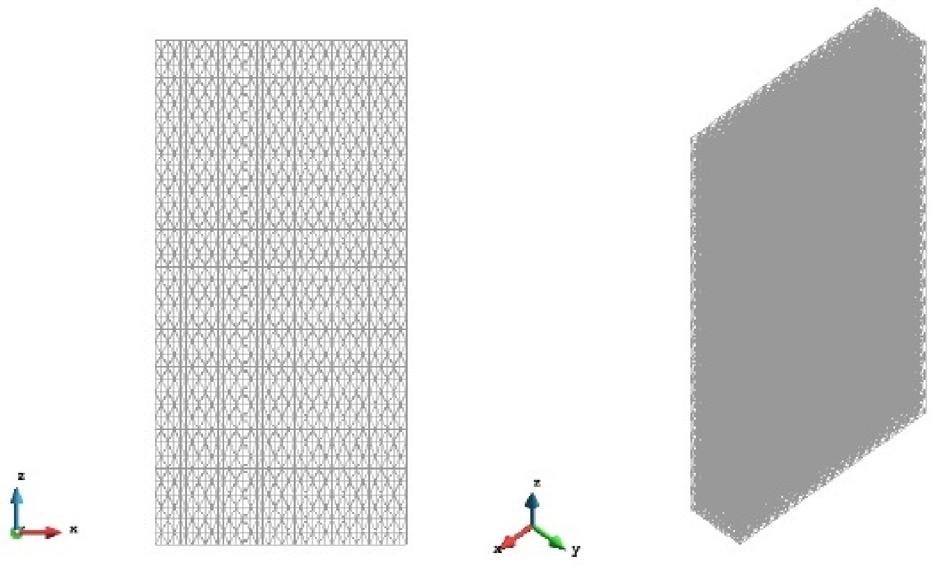


Figure 4: Box meshed with tetrahedral elements

Analysis settings

This section describes the definition of parameters required to average the DEM results using P4. This includes the specification of which coarse graining function to be used, coarse graining width and cut-off factor (for more details refer P4 manual), the method of time averaging and format of output file for saving the data.

Click on the problem data icon of P4 toolbar as shown in Fig 2 which opens up a problem data window.

The following values were used for this exercise:

Input

- `PARTICLES FILENAME` : Read the .p4p file (refer to P4 manual for conversion of DEM data to .p4p)
- Tick Process Contacts Files option.
- `READING TIME STEP` : ALL
- `STEP FREQUENCY` : 1

Time Averaging

- Activate Time Averaging is checked.
- `TIME AVERAGE` : ALL.

Spatial Averaging

- `TYPE` : COARSE GRAINING
- `FUNCTION TYPE` : GAUSSIAN
- `WIDTH` : GLOBAL
- `WIDTH VALUE` : 0.0045
- `CUT-OFF VALUE` : 3
- `SPATIAL INTEGRAL` : NO.

- `CALCULATE GRADIENTS` : YES.

Output

- `FILE TYPE` : GiD Binary
- `OUTPUT FILE` : Other
- `OUTPUT FILENAME` : Give appropriate output file. P4 writes the all data to this file.

Click on start process as shown in Fig 2 and the progress can be monitored by clicking on information icon next to start process icon. A message pops up on the screen after the process is finished. This generates .bin file in the working directory. At this stage, the pre-processing of DEM results is finished and by clicking on the post processing icon shown in Fig 2 the GiD switches to post processing mode.

Post-processing

This section describes on how to visualise the results using contour plots and graphs. GiD has also several other options to visualising the results, for example iso-surfaces, streamlines etc (refer GiD manual for more information).

Load result file

Select the result that was generated by clicking on the icon as shown in the Fig.5 and this loads the result file onto the mesh that is created in pre-processing.

Select the time step for visualisation

You can select the time step at which results has to be visualised. As in this case, the time averaging has been done for all time steps, only the results at averaged time step is available. `View Results ⇒ Default Analysis/Time step ⇒ Time Step`. Select the time step for analysing the result. In this case only one time step is displayed.

Select display style

There are various options in GiD to display the geometry which are discussed in P4 Manual. The display can be customised, for example turning off the mesh elements, turning on/off the other geometric elements. The display of mesh elements and lines that are created for graph can be turned off from the *SELECT AND DISPLAY STYLE* window. The geometric entities present in the domain are displayed as a list in *SELECT AND DISPLAY STYLE WINDOW* and any selected entity can be turned off from display by click on I/O. Similarly the mesh elements can be turned off by selecting the solid option for the section ST as shown in Fig 5.

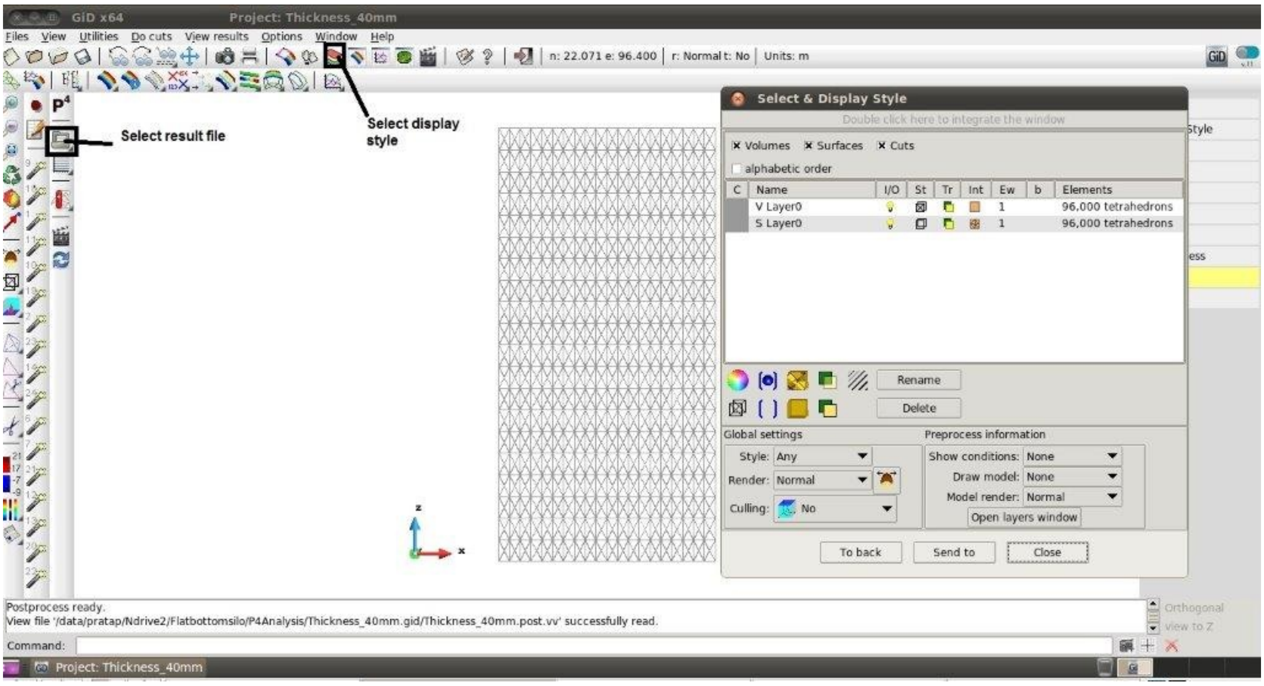


Figure 5: Layout of GiD post processor

Visualisation and export contour plots

The flow patterns at selected time step can be visualised as contour plots of velocity as illustrated below:

- `View Results ⇒ Contour Fill ⇒ Velocity ⇒ Velocity Z`
- `View Results ⇒ Contour Fill ⇒ Velocity ⇒ Velocity X`

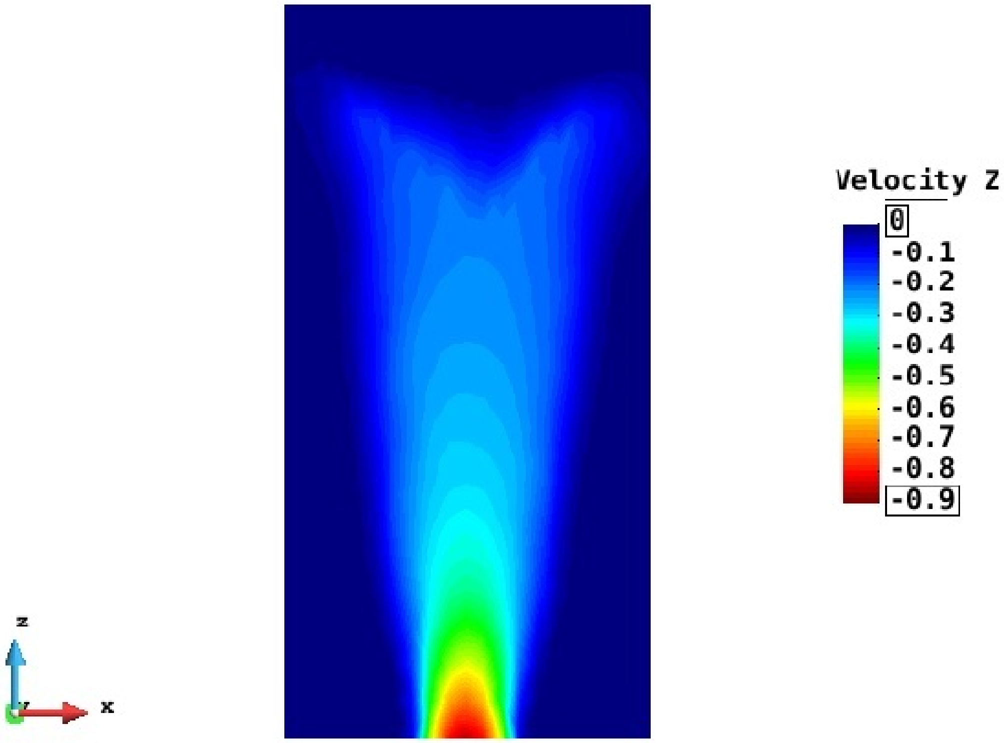


Figure 6: Contour plot of vertical velocity (m/s)

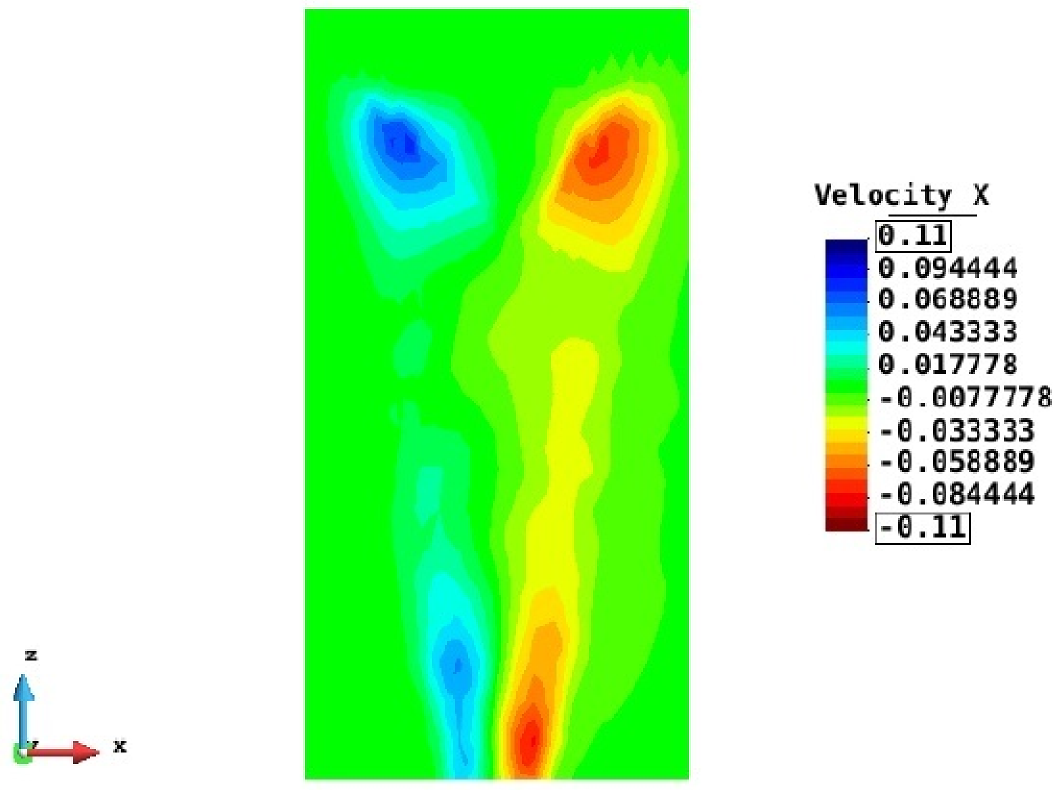


Figure 7: Contour plot of horizontal velocity (m/s)

The images can be saved as .tiff format by selecting `File ⇒ Print to file ⇒ Tiff`. The page settings (size, resolution) for exporting the image can be changed from `Files ⇒ Page and Capture Settings`.

Plot and exporting graphs Similar to the creation of lines as described in section 2.2, macros can be recorded for plotting a single graph for a given variable on given line along specified direction. The macro can be edited create multiple graphs at multiple location on a single click. The procedure for this is as follows:

1. Start record macro (as mentioned in section 2.2)
2. `View Results ⇒ Line graphs ⇒ Set X axis ⇒ X variation`
3. `View Results ⇒ Line graphs ⇒ Velocity ⇒ Velocity Z`
4. Input the coordinates (-0.075, 0.015,0 and 0.075, 0.015, 0) of the line through command window. This generates a graph.
5. `File ⇒ Export ⇒ Graph ⇒ All`. This saves the graph as ASCII file which can read into MATLAB or EXCEL.
6. Stop record the macro.
7. Edit the created macro (displayed next to record macro button) by right clicking on it and select Edit window. Copy paste the first command into subsequent lines and change the line definition and the variable name to be plotted.

The vertical velocity profiles plotted at five different Z locations and at Y= 0 is shown in Fig 8. The graph is produced from MATLAB by reading ASCII file generated by running the above created macro.

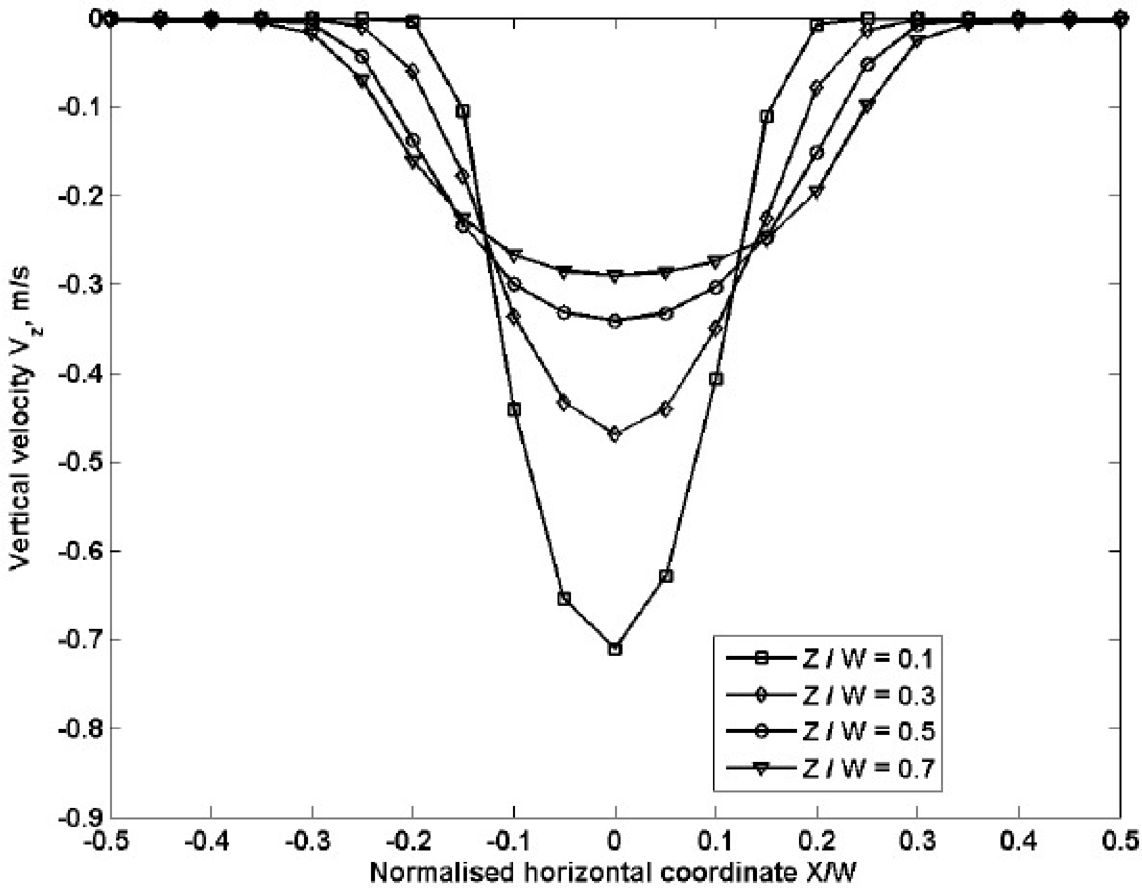


Figure 8: Vertical velocity profiles at different heights from the outlet.

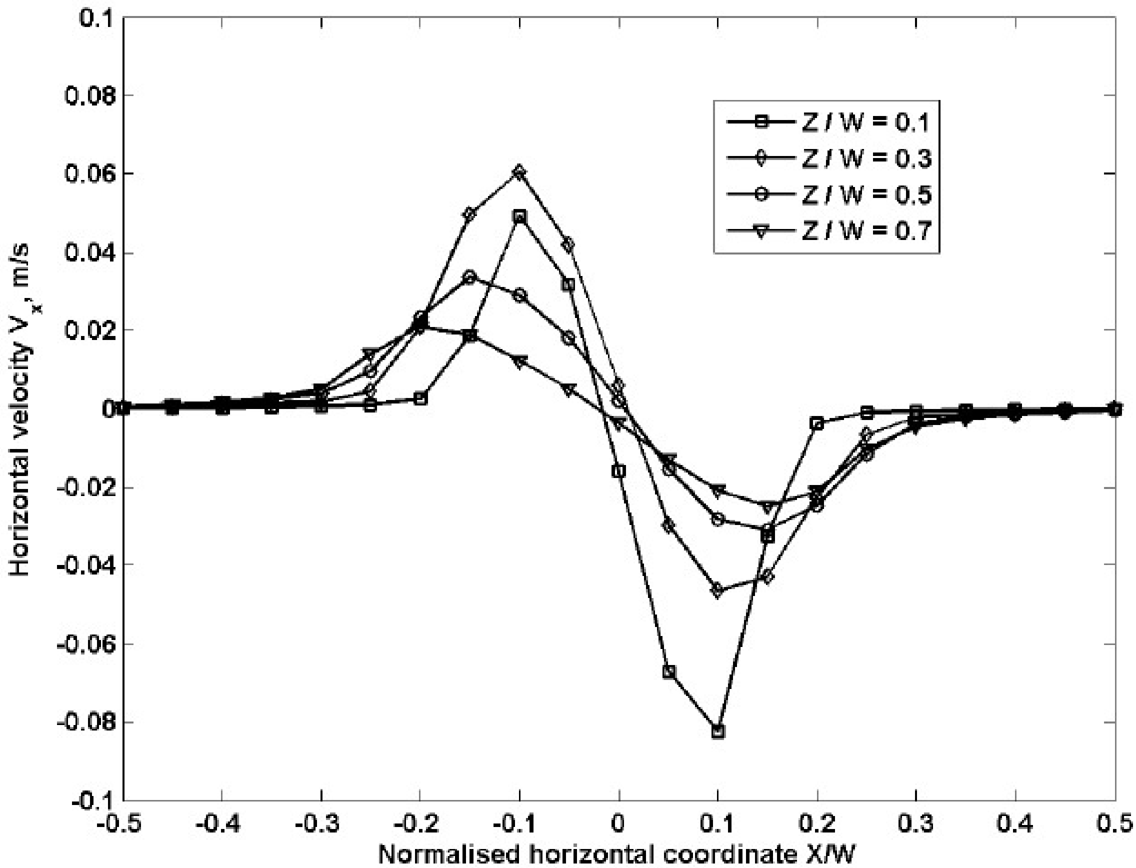


Figure 9: Horizontal velocity profiles at different heights from the outlet.