University of Exeter coursework header sheet

ECMM102 Group Project (Meng) (A, TRM1+2 2014/5)

031308

1015549

610006687

Coursework: I2

Submission Deadline: Thu 30th Apr 2015 12:00

Personal tutor: Dr Christopher Smith

Marker name:	Gavin Tabor
Word count:	12087

First marker's comments	Indicative mark
Second marker's comments	Second mark
Moderator's comments	Agreed mark

I2 Report

Inlet Conditions for Large Eddy Simulation
of the Atmospheric Boundary Layer
James Richmond

2015
4th year MEng Group Project

I certify that all material in this thesis that is not my own work has been identified and that no material has been included for which a degree has previously been conferred on me.

Signed...

College of Engineering, Mathematics, and Physical Sciences
University of Exeter

I2 Report

ECMM102

Title: Inlet Conditions for Large Eddy Simulation of the Atmospheric Boundary Layer
Word count: 12087
Number of pages: 38

Date of submission: Wednesday, 29 April 2015

Student Name: James Richmond
Programme: MEng Mechanical Engineering
Student number: 610006687
Candidate number: 031308

Supervisor: Gavin Tabor

Abstract

Large Eddy Simulation (LES) is a subset of fluid simulation in which turbulent behaviour is modelled explicitly instead of approximated by a turbulence model. LES is useful for evaluating the dynamics of turbulent flows such as vortex shedding. In order for LES to accurately model turbulent behaviour, turbulence must be explicitly defined at the inlet. A good inlet condition can cut down on the size of domain required and can increase the accuracy of the solution by presenting a more physical initialisation than a basic random approach.

The aim of this project is to research and develop an implementation of the Synthetic Eddy Method (SEM) for use within an OpenFOAM solver. This method involves the definition of a number of eddies within the flow and simulates them accordingly. By a combination of the eddy locations and intensities along with the values of Reynolds stress throughout the mesh a coherent turbulence field is generated. This approach is well suited for simulating the turbulence within the atmospheric boundary layer.

The inlet condition is validated against the random noise inlet condition included in OpenFOAM by default along with a basic sinusoidal inlet. The synthetic eddy method is shown to provide a suitable inlet condition generating physical turbulence with a good degree of numerical accuracy. Additionally, the implementation of the synthetic eddy method is discussed in terms of practical concerns relevant to the test domain.

Keywords: Synthetic Eddy Method, Large Eddy Simulation, Inlet Turbulence

Table of contents

1. Introduction and background

Atmospheric air flow is a complex area of computational fluid dynamics due to the numerous factors affecting it. The large scale air flows within the atmosphere are often highly turbulent and vary dramatically with time. As such traditional approaches to solving computational fluid problems are not suitable since they work best on small scale flows where turbulence can be approximated by a time average. As such the best approach to the simulation of the atmospheric boundary layer is using a method referred to as Large Eddy Simulation (LES) and derived methods. This approach significantly reduces the scale on which turbulence is approximated allowing the larger turbulent effects to be resolved explicitly while maintaining somewhat reasonable run-times. This approach does therefore require the large scale turbulent effects to be present at the inlet of the domain which can be problematic.

A number of approaches exist to the creation of turbulent inlets ranging from the overly simplistic (random noise) to mathematically complex (wavelet superposition). Generally speaking the more complex methods are able to create more realistic turbulence patterns but in the absence of a total understanding of turbulent behaviour, all current methods are approximations.

The Synthetic Eddy Method (SEM) is an approach to the creation of turbulence based on the simulation of eddies within the flow of a characteristic length scale. This approach mimics the generally accepted model of turbulence as a superposition of coherent structures giving it a qualitative benefit over other comparable methods. Also, since this project is concerned from atmospheric air flow, a method based on the Reynolds Stress such as SEM would be better able to model turbulence due to the presence of shear layers in the mean velocity profile. The Synthetic Eddy Method has been researched and developed over several years but is not yet generally accessible. The goal of this project is to create and evaluate an SEM model for use with OpenFOAM with regard to atmospheric air flow.

The aims of this project are encompassed by a larger group project on the aero-elastic properties of a bridge deck structure. By the creation of a turbulent inlet condition capable of accurately representing atmospheric flow, a high quality CFD simulation can be performed on a bridge deck mesh. Some results pertinent to this report gathered from other group members are attributed accordingly and discussed in the relevant section.

1.1. *Aims and Objectives*

The aim of this project is to create an implementation of the Synthetic Eddy Method using the OpenFOAM class library. This model will then be compared to other approaches of generating turbulence at the inlet. By the creation of this generalised inlet turbulence generation method, the aims of the group project can also be aided by applying the method to a model of the atmospheric boundary layer. This aim is broken down into a number of objectives listed below.

- Research into the synthetic eddy method and other turbulent inlet generation methods
- Create an implementation of the synthetic eddy method for use within OpenFOAM
- Test the method against other turbulence generation schemes for validation and comparison purposes
- Define an inlet condition for atmospheric flow for use within the group project

2. Literature review

The prior literature relevant to this project covers a few different areas. Primarily work on inlet conditions for large eddy simulation will be drawn on with special regard to those using the Synthetic Eddy Method. Additionally research on the definition will be used when configuring the inlet condition for use with the atmospheric boundary layer specifically.

2.1. *Large Eddy Simulation*

Large eddy simulation is a method of fluid simulation which increases the amount to which the flow is explicitly solved. An overview to the method is presented by Moeng [1]. The principal of the method is instead of using a time averaging approximation to obtain the solvable flow; a spatial averaging scheme is used. Anything flow that exists on a scale above the grid scale is solved for explicitly while anything below it is approximated. For this approach to be numerically accurate, any LES simulation must therefore be resolved down to the smallest turbulent length scale. This introduces a strong coupling between cell size and Reynolds number as described by Charles and Sagaut [2]. These mesh requirements typically mean an LES simulation has a significantly larger number of cells than a Reynolds Averaged

Navier-Stokes (RANS) simulation of the same domain. The benefit of using LES over a standard RANS simulation is that the turbulent flow can be derived, which is especially useful if the transient effects are of interest. Because of this, LES is commonly used for large scale atmospheric flows and in simulations where vortices within the flow are expected and relevant to the solution.

A requirement of a simulation using an LES turbulence model is that turbulence must be defined explicitly at the inlet down to the grid scale. Much research into inlet conditions has been done attempting to improve the quality of the turbulence generated. Various methods for the generation of turbulence at the inlet are discussed by Tabor and Baba-Ahmadi [3]. The general requirements of any inlet condition are to appear random, exist on length and temporal scales relating to the problem and simulation, appear suitably physical relative to the Navier-Stokes equations and be practical to use and adjust within the setup of a simulation. This paper mentions a number of known methods of inlet turbulence generation: Random Noise, Fourier techniques [4], noise filtering [5], precursor simulation [6], input of experimental data utilising Proper Orthogonal Decomposition (POD) methods [7] and synthetic eddy simulation [8].

Some of these methods can be ruled out for this project due to unsuitability. Due to the scale of the atmospheric boundary layer any experimental procedure below full scale would be unlikely to be able to represent turbulent fluctuations on all length scales. In any case, the collection of experimental data for this case is a significant undertaking and beyond the scope of this project. The approaches of using precursor simulation and POD methods require simulations to be run capable of generating a full representation of the turbulent patterns. Since this project is concerned with atmospheric flow this approach is not viable due to its highly turbulent nature. Such as simulation would have significant computational cost and take several weeks to fully represent the problem. As such an estimation of the turbulence patterns better suits the projects scope. Fourier methods require the definition of a number of frequencies and amplitudes which relies heavily on mathematical theory and experimental data whereas the synthetic eddy method builds this from a turbulent length scale derived arithmetically. Due to the difficulty of simulating and experimentally recording atmospheric air flow, the latter approach will be researched. Since there is no current publically available implementation of the synthetic eddy method within OpenFOAM, the creation of one is a suitable goal for this project.

2.2. Turbulence

One method for generation of synthetic turbulence by simulating eddies was proposed by Jarrin et al. [8]. The implementation of the synthetic eddy method designed for this project is based on the one described by Jarrin in this paper and some of his subsequent research papers on the method [9] [10]. The concept of the simulation of eddies in the flow to create a turbulence profile is a more physical approach than other methods relying on numerical patterns such as wavelet and sinusoidal inlet conditions. This method is easily scalable by the number and size of eddies and therefore does not require a significantly refined mesh to generate an accurate result. Due to the limited scope of this project, any method with a high computational cost would be difficult to create and verify within the timescale available. The above papers will be the primary reference for design and validation of the inlet condition created during this project. A more in-depth review of the method's implementation is shown later in this report.

Recent developments in the field of Large Eddy Simulation involve new ways of generating inlet turbulence efficiently and accurately with minimal computational cost. Muñoz-Esparza et al. [11] describe a novel method of turbulence generation for specific use with simulation of the atmospheric boundary layer. The method described in the paper does not require a precursor simulation and therefore is more computationally efficient than precursor simulation methods including SEM. However, both the approach in this paper and the SEM approach are computationally cheaper than the old approach of taking inlet data from an inlet domain fully simulated using LES. The method described by this paper uses a cell-based quasi-random function to generate inlet characteristics. An approach such as this will gain accuracy as the number of cells increases as the scale at which turbulent characteristics can be accounted for decreases towards its minimum. However, on larger grid scales the fluctuations below the grid scale may be lost. This is a benefit of the SEM method since the separation of the eddies definition from the mesh definition allows all length scales to be accounted for regardless of mesh definition.

2.1. OpenFOAM

Throughout this project, the open source package OpenFOAM [12] will be used for simulation of test domains and the creation of the inlet condition. OpenFOAM is a C++ class library accompanying a series of pre-made solver applications capable of solving the vast majority of fluids problems. Due to the accessibility of the source code OpenFOAM is also a viable choice for the creation of custom inlet conditions such as the one described in this report.

The inlet condition described by this report is compared against other inlet specifications within the OpenFOAM library. These include an inlet utilising the built in *turbulentInlet* random noise inlet and a sinusoidal inlet. The external library swak4Foam [13] is used to allow the custom creation of a sinusoidal inlet condition. This library contains the boundary condition *groovyBC* which allows the specification of a boundary patch's behaviour by an arbitrary script. This is used primary for the creation of the sinusoidal inlets listed in this report due to ease of use compared to creating an inlet condition within OpenFOAM. This same approach is not possible for the SEM inlet since it requires much closer control of its implementation.

2.2. Atmospheric Boundary Layer

The characteristics of turbulence within a flow is highly dependent on the nature of a flow. Turbulence generated by channel flow may not be suitably representative of turbulence within the atmospheric boundary layer. To properly create a turbulent inlet for atmospheric flows, the simulation of such flows with computational methods is considered.

An overview of the nature of the atmospheric boundary layer (ABL) with regards to fluid flow is presented by Garratt [14]. The ABL is described as the height of the atmosphere in which the air flow is significantly affected by the terrain below it and upwind of it. As such, the height of the ABL can vary significantly within a region. Simulation of the entire height of the ABL would involve tracking temperature and buoyancy effects, calculating the Coriolis force and measuring the humidity throughout the domain. Such a simulation is very costly computationally and would not yield useful results to the overall project. As such, an approximation to the velocity profile will be taken up to a limited height. For the purpose of this project and the associated group project, this mean profile will be generated by a study of the surrounding terrain by another group member.

Mason [15] describes the set-up and simulation of the convective atmospheric boundary layer using LES approaches. Specifically the Smagorinsky turbulence model was chosen and was shown to be a suitable choice for the simulation given some tuning of the model's variables. This paper gives an indication to the types of results which can be expected in a similar simulation. The profiles shown in the paper are highly turbulent and clearly show the presence of large scale eddies. This is illustrative of the importance of inlet conditions in atmospheric flow simulations.

Computational simulation of the atmosphere requires a large number of computational cells and the operation of solvers capable of dealing with simulation of this scale. OpenFOAM has been used in the past for simulation of the atmospheric boundary layer as shown by Lignarolo et al. [16]. Since it is known that the solver is capable of handling the large scale flows present in the atmospheric boundary layer, it is suitable for the scope of this project.

3. Theoretical background and design

The project described by this paper involves design of inlet conditions in a manner consistent with the computational solver's specification. As such, an understanding of the operation of computational solvers is required to accurately define the inlet condition. Since the approaches described need to be adapted to atmospheric air flow, the nature of atmospheric air flow is also discussed. The setup of a validation case and the factors relevant to quantifying the results are also discussed in this section.

3.1. Computational Fluids Simulation

The mechanics of fluid flows can be evaluated by considering the Navier-Stokes equations, a set of governing equations generalised to all fluid flows. Additional terms to those shown in equations 1 can be added to account for problem-specific effects such as sources, sinks, Coriolis forces and ferromagnetism.

Equation 1: $\rho\left[\frac{\partial u}{\partial x} + (u \cdot \nabla u)\right] = -\nabla P + \mu \nabla^2 u$

Equation 2: $\nabla \cdot u = 0$

The Navier-Stokes equation is a continuum equation consistent to all parts of the flow on all scales. Most computers are not suited to the solving of continuum problems, the equations are discretised to evaluate the equation between a large number of finite volumes which when combined represent the problem domain. These finite volumes are typically arranged in a grid and collectively referred to as a mesh. An inlet condition is to be created defining the velocity of a number of cells collectively forming an inlet face. Such a definition must be representative of the physical behaviour of fluids as described by the Navier-Stokes equation as well as maintain continuity of incompressible flow.

3.2. Turbulence

Turbulence within a flow can be characterised by evaluating Reynolds number. Reynolds number is a dimensionless number relating a characteristic length scale, characteristic velocity and fluid viscosity to indicate the turbidity of a flow shown in equation 3.

Equation 3: $Re = UL/\nu$ *where U is a characteristic velocity, L is a characteristic length and ν is the kinematic viscosity of the fluid.*

A flow with a Reynolds number below 2000 is considered laminar and contains little to no turbulent effects. Reynolds numbers above 2000 are considered turbulent and require the turbulence to be accounted for in their evaluation. The higher Reynolds number is, the larger the degree of turbulence in the flow. Atmospheric flow for example typically has Reynolds numbers in the millions with the exact value depending on localised characteristics.

Turbulence is often thought of as a superposition of coherent structures called eddies superimposed on top of a mean (time invariant) flow. These structures can exist on a range of scales from the largest limited only by the shape of the domain to the smallest on the Kolmogorov length scale [17] where smaller structures dissipate into heat. Larger eddies break down with time into a number of smaller eddies creating a cascading distribution of length scales. Resolution of all these eddies within a flow is not currently possible as they are too numerous to simulate within any reasonable time-scale and their numbers increase with

Reynolds number. Instead, they are approximated in a number of ways to allow computation of more generalised flow characteristics.

3.3. Reynolds Averaged Navier-Stokes Equations

The simplest approach to solving a turbulent fluid flow is to take a Reynolds average of the flow to separate it into a mean flow invariant of time and a turbulent flow overlaid onto it. The mean flow can simply be resolved by application of the discretised Navier-Stokes equations which only leaves the turbulent flow to be solved. This is problematic since the solution is non-linear and leads to the creation of a Reynolds stress term shown in equation 4.

Equation 4: $$R_{ij} = \rho \begin{bmatrix} {u'_x}^2 & u'_x u'_y & u'_x u'_z \\ u'_y u'_x & {u'_y}^2 & u'_y u'_z \\ u'_z u'_x & u'_z u'_y & {u'_z}^2 \end{bmatrix}$$

The Reynolds stress term is closely related to the turbulence of the flow since it characterises the interaction of the turbulent flow in each direction within a given computational cell. This value is unknown at the time of simulation and therefore must be approximated by a chosen turbulence model. Several commonly used turbulence models exist and are used to approximate this value to a turbulent length scale.

The Reynolds stress term can be useful when defining inlet conditions as will be seen later in this report. The actual value of the Reynolds stress used in those inlet calculations will be derived from the values associated with the turbulence model. As such, any error within the turbulence model will be present within the Reynolds stress term used for the SEM inlet. This places a significance on the numerical accuracy of the inlet domain used for the generation of the inlet condition.

3.4. Large Eddy Simulation

In some situations, the turbulence within the flow is crucial to generating an accurate solution. A common example of this is evaluating situations where vortices are shed causing flow oscillations. Simulations using Reynolds Averaged Navier-Stokes (RANS) turbulence models often overestimate the degree of turbulence within these vortices [18]. The approach of Large Eddy Simulation (LES) is to use a spatial averaging scheme as opposed to the time averaging scheme used by RANS methods. Whereas RANS methods approximate all time-variant structures, LES methods only approximate those which are below a given length scale (typically the grid scale). All turbulent flows above the length scale are solved alongside the mean flow to account for the larger turbulent effects explicitly.

Within RANS methods, the turbulent properties of the flow moving into a domain can be easily accounted by equations relating the turbulent properties to known physical quantities such as Reynolds number and mean flow velocity. However, LES methods require those turbulent fluctuations above the spatial filter's scale to be presented explicitly at the inlet of the domain. To do so in a physical manner is an active area of research and no current method is currently considered entirely correct. A number of methods for inlet turbulence specification are discussed in the sections below.

All these inlets are required to meet a set of criteria to be suitable as discussed in the literature review previously. They must obey continuity in that the turbulent fluctuations cannot influence the mean flow when time-averaged. They should be random in nature since any structured flow is more representative of a time-varying inlet than transient turbulence. The turbulence should aim to represent a series of coherent structures both for physicality and to allow their preservation from averaging due to viscosity effects in the fluid simulation.

3.4.1. Random Noise

Random noise is simple to generate and can easily be guaranteed to have a known mean and distribution around it. By introducing filtering methods, more coherent structures can be created from it both spatially and temporally allowing the introduction of a turbulent length scale. Random noise is essentially the baseline of any turbulent inlet since it meets the practical requirements without any significant degree of physicality. Any further methods should aim to produce results comparable to a random noise approach with better representing the range of frequencies and length scales present in physical turbulence. The random-noise based inlet present with OpenFOAM '*turbulentInlet*' uses a degree of filtering to preserve some spatial coherence since it would otherwise be averaged away almost immediately.

3.4.2. Sinusoidal Turbulence & Fourier Methods

The principal of Fourier synthesis states that any periodic signal can be represented by a number of sinusoids of given amplitude, phase and frequency. Therefore, it is theoretically possible to represent a turbulent flow's characteristics by a number of sinusoids superimposed over a mean flow. The structure of sinusoids lend themselves to the problem since they are already defined by frequencies, amplitudes and phases similar to eddies with associated length scales, intensities and positions. The definition of a series of sinusoids allows for scalability of the inlet's complexity as more sinusoids are superimposed. Continuity is assured by the fact that sinusoids have zero mean assuming the largest wavelength is entirely contained within the domain. Fourier methods are a complex area of research highly dependent on mathematical theory. For this report, a simplistic model of sinusoidal turbulence is described for comparison to the selected inlet turbulence generation method.

A subset of Fourier synthesis worth noting is wavelet synthesis which replicates a non-periodic signal by the superposition of sinusoidal structures with limited influence. These structures more closely resemble the description of eddies and are better able to represent the temporal characteristics of individual eddies than sinusoids with influence over a large area.

3.4.3. Proper Orthogonal Decomposition & Precursor Simulation

Proper Orthogonal Decomposition (POD) is more of an analysis tool than an inlet specification. By evaluating experimental flows of the results from the outlet of an experimental case or a precursor simulation, POD methods can infer generalised flow characteristics. This reduction of flow characteristics from the mesh scale to a series of quantities allows other methods to be better initialised using quantities more representative of the expected flow.

While precursor simulations can be used to gain insight into the qualities of a flow such that another inlet type can be specified, in some cases it is better to use the precursor simulation directly. Given enough time and space the turbulence present within an inlet will develop into a more physical form of turbulence due to the progressive application of the Navier-Stokes equations. Such precursor simulations are computationally costly since they can take as long as the simulations they are to provide inlet conditions for. The benefits of such approaches are that the simulation residuals are guaranteed to be low since they have already been simulated and they retain a degree of physicality relative to the flows mechanics. Precursor simulations and associated inlet data can be used in successive test cases lowering the overall cost as it is distributed across many runs.

3.5. Synthetic Eddy Method

The main part of this project involves the creation of an implementation of the Synthetic Eddy Method (SEM) for use within OpenFOAM. This approach involves the definition of a sub-region within a domain in which eddies are transported through and allowed to manipulate the velocity field. The aim of using this method is to create a turbulent velocity field that resembles a number of coherent eddies overlaid onto a mean flow. A detailed breakdown of the approach used is detailed below.

For the purpose of this section an eddy will be described as a structure within the flow existing at a given position, with a given intensity and a turbulent length scale. The length scale chosen has a significant effect on the nature of the turbulence generated. In practical terms it represents the distance at which a cell can be influenced by a given eddy which when coupled with a given number of eddies represents the turbulence of a given flow field. The eddy length scale is often chosen based on the turbulent characteristics of the flow. However, it is not allowed to drop below the grid scale since that introduces the possibility of a given eddy being unable to affect any cells on a given time step and therefore removing the continuity of the method. A general formula for finding a reasonable length scale σ is shown below in equation 5. The eddy length scale can be set per-eddy or per cell within the domain among other methods. For this project the length scale will be specified uniformly based on average flow characteristics.

Equation 5: $\sigma = \max\left(min\left(\frac{k^{\frac{3}{2}}}{\epsilon}, \kappa\delta\right), \Delta\right)$

The sub-domain in which eddies are simulated is specified by the user. This volume ca be any arbitrary shape but should be specified in a contiguous manner to ensure continuity of eddies passing through the domain. Eddies are allowed to exist within this domain and partially outside it as long as a cell exists closer than the eddy length scale. If an eddy moves out of range, it is removed and replaced by a new random eddy placed on a face on the opposite side of the domain. This ensures a constant number of eddies while maintaining a statistically even distribution of turbulence. For the base implementation of this method, a box aligned with the primary axis is used.

The number of eddies required in the domain depends on the eddy length scale chosen and the size of domain. Enough eddies need to be introduced such that there is 'statistical coverage' of the entire domain. This can easily be guaranteed if the eddies are arranged in a uniform grid but often a larger number of eddies are required since a random positioning scheme is chosen. A rough estimate of the number of eddies required can be gathered using the below formula.

Equation 6: $N_{eddies} = \frac{V_B}{\sigma^3}$ *where* V_B *is the volume of the box of eddies*

Each time step the eddies are transported through the box of eddies by the mean flow. An alternative approach would be to use the local velocity to move the eddy but this can introduce problems if the eddy meets an irregular flow such as by a wall or in a recirculation zone. In these situations, errors are more likely to occur as the flow field is repeatedly influenced by an eddy in the same position. To reduce the likelihood of any of these errors occurring, all the eddies within the box will be moved by the same velocity which is averaged throughout the box of eddies prior to running.

At each cell within the box of eddies, a modified velocity signal is calculated taking into account the eddies surrounding the point. This can be done independently for each cell in the mesh since the contribution of the surrounding flow is already taken into account in the Reynolds Stress tensor included within the formula. The general equation for the modified velocity at a given cell is shown in the equation below.

Equation 7: $u_i = U_i + \frac{1}{\sqrt{N}} \sum_{k=1}^{N} a_{ij} \epsilon_j^k f_{\sigma(x)}(\boldsymbol{x} - \boldsymbol{x}^k)$

Where U_i is the mean flow velocity, N is the total number of eddies, ϵ_j^k is the eddy's intensity and a_{ij} is the cholesky decomposition of the Reynolds Stress tensor as shown below.

Equation 8: $$a_{ij} = \begin{pmatrix} \sqrt{R_{11}} & 0 & 0 \\ R_{21}/a_{11} & \sqrt{R_{22} - a_{21}^2} & 0 \\ R_{31}/a_{11} & (R_{32} - a_{21}a_{31})/a_{22} & \sqrt{R_{33} - a_{31}^2 - a_{32}^2} \end{pmatrix}$$

The function $f_{\sigma(x)}(\boldsymbol{x} - \boldsymbol{x}^k)$ is the velocity distribution which defines the effect of an eddy on a cell based on the difference between the cell's location $\boldsymbol{x}$ and the eddy's location $\boldsymbol{x}^k$. This function is heavily dependent on the eddy length scale chosen along with the size of the box of eddies. The velocity distribution formula is shown below in equation 9.

Equation 9: $f_{\sigma(x)}(\boldsymbol{x} - \boldsymbol{x}^k) = \sqrt{V_B}\sigma^3 f\left(\frac{x-x^k}{\sigma}\right) f\left(\frac{y-y^k}{\sigma}\right) f\left(\frac{z-z^k}{\sigma}\right)$

The function $f\left(\frac{x_i - x_i^k}{\sigma}\right)$ is referred to as the shape function. The exact specification of this formula is open to interpretation and can depend significantly on the case being simulated. When coupled with the choice of length scale the shape function can indicate the distribution of eddies at different scales throughout the flow and the energy of the eddies at these scales.

The shape function is to take a value of zero when the distance between the cell and the respective eddy is greater than the eddy length scale. Since this can be programed in as a prior condition, the shape function only needs to be physically representative within the range $[-\sigma, \sigma]$. The function must also meet the criteria of having the normalisation $\|f\|_2 = 1$.

For the sake of simplicity a basic tent function is used $f(x) = \sqrt{\frac{3}{2}}(1 - |x|)$ such that there is a linear relationship between the distance between the eddy and the cell with the maximum value taken when directly next to the eddy.

Since the Reynolds stress tensor is symmetric the fluctuations generated by the cholesky decomposition are effectively directionless. The eddy's intensity is taken into account to arbitrarily specify the direction the stress is acting. The exact specification of eddy intensity is open to interpretation but the assignment of a 1 of -1 value by a uniform random distribution representing each axis direction has shown to be sufficiently accurate by prior publications [8].

3.6. Atmospheric Boundary Layer

Since this project is ultimately targeted to generating an inlet profile for use with atmospheric air flow, some knowledge of the atmospheric boundary layer is required. The atmospheric boundary layer (ABL) is by definition the height within the atmosphere at which the air flow is affected by the ground. This height can vary dramatically across terrain due to a number of factors but is typically considered to be around 800-1000m in most cases. Above this there are macroscopic driving flows driven predominantly by the Coriolis force. On these large scales the ABL becomes very complex and requires a large number of factors to simulate fully: temperature, humidity, gravity, local terrain, current wind conditions, etc.

In order to limit the computational complexity the ABL is restricted to the area of interest and restricted to averaged characteristics. To focus on the group project this report contributes toward, the ABL will only be considered up to the height of around 100m. This will provide a reasonably accurate profile for the subject of the group project (a bridge deck at the height of 30m). While the exact profile will be provided by another group member the characteristics of such a profile should be considered when designing a representative turbulent inlet.

Atmospheric air flow is highly turbulent and typically has a Reynolds number of 1-10 million depending on local conditions. Taking the characteristic length as the height of a bridge deck and an air speed of 5m/s the derivation of the local Reynolds number can be seen in Equation 10 below.

Equation 10: $Re = \frac{UL}{\nu} = \frac{5\times4.6}{1.51\times10^{-5}} = 1{,}523{,}178$

On top of the high Reynolds number, ABL flow can be characterised as a shear flow since the horizontal velocity changes significantly with height while vertical motion is generally minimal. Close to the ground the velocity is low, increasing to a maximum value at a decreasing rate synonymous with typical boundary layer behaviour but on a larger scale.

Considering these factors in terms of inlet turbulence a number of conclusions as to the approach can be drawn. Since the velocity profile is not uniform, the turbulence should not be uniform either. The shearing flow close to the ground will cause the highest degree of turbulence and the inlet turbulence model should represent that.

3.7. Experimental Procedure

The overarching aim was the implementation of the Synthetic Eddy Method within OpenFOAM which involved programming a number of files to be compiled into an existing OpenFOAM solver, namely *pisoFoam*. Using the *groovyBC* boundary condition from the Swak4Foam library, a sinusoidal inlet was created for comparison to the SEM inlet. Additionally, a case was created for the validation of the various models such that they can be directly compared.

3.7.1. Synthetic Eddy Method Inlet Creation

The creation of the synthetic eddy inlet condition progressed over a large duration of the project. Originally the method was considered as a method of perturbing the flow each time step in a physical manner. As such, it was created as a solver with conditions set up to limit its operation to a sub-domain by the inlet. This method was faulted on the grounds of numerical error which caused compound errors in successive time steps. This problem is inherent in the method since for the result to remain physical the magnitude of the velocity at a given point should not change after perturbation. The model is able to maintain this constraint to a high tolerance on a single iteration but any model inaccuracy whatsoever will lead to eventual failure since the method is not self-stabilising.

In order to prevent the compounding of errors on successive time steps the solution was to decouple the mean velocity field and the perturbed field. The SEM solver was run on a domain using a RANS turbulence model to unsure the stored velocity represents only the mean flow. Using this mean flow and the characteristics derived from it a perturbed field representing the turbulent fluctuations was written to a secondary field representing the combination of the mean flow and the perturbations around it. By preventing the perturbations from feeding back into the RANS simulation, the flow remained stable while providing a unique perturbed field in each time step. Use of the OpenFOAM utility *sample* allowed the perturbed field data to be extracted from each time step and converted into a format for use with the *timeVaryingMappedFixedValue* patch as an inlet condition for a secondary LES simulation.

This approach has several benefits and drawbacks associated with it over encapsulating the method in an inlet conditions. By creating the perturbed field data in a precursor simulation the same turbulent field data can be applied to a number of different cases. This effectively creates a simulation overhead relating to the time required to create enough inlet data compared to a slowdown on each time step for a simulation using a related inlet condition. Therefore, if you plan on using the inlet data more than once it saves computation time to generate it in advance. Also, since the inlet data can be decoupled from the test domain the case generating the inlet data does not need to include any complex geometry relating to the simulation and as such will run faster.

By allowing the data to be used by an OpenFOAM maintained condition the secondary simulation is not limited to using a specific solver or turbulence model. This also simplifies the process of mesh decomposition and parallel simulation. However, since the inlet data is required in advance, there is a large memory footprint of the inlet condition over an inlet condition which generates the field data on a per time step basis.

By using the method as implemented in a solver it is possible to use the method for field initialisation. It is possible to run the solver for a single time step on any mean flow simulation of the test domain to create physical turbulent fluctuations throughout the entire domain. This form of field initialisation can save a significant amount of time required for an LES simulation to reach a stable turbulent state.

3.7.2. Atmospheric Velocity Profile

The mean atmospheric velocity profile is generated from an analysis of the terrain surrounding an area of interest using a mathematical approximation as a baseline. This analysis will be completed by another member of the group project who is studying terrain effects. This profile is then approximated to a mathematic function using statistical analysis such that is can be easily be adapted to any domain chosen.

The atmospheric boundary layer is a shearing flow and there is little movement in the vertical direction compared to the mean flow. As such, the mean atmospheric velocity profile will be approximated to have no vertical motion to simplify its specification within the boundary condition. The flow direction in the horizontal plane is kept constant since the Coriolis force does not affect the flow significantly within the height of domain used for testing. Macroscopic effects such as the influence of the jet stream and prevailing wind are ignored to generalise the solution and approach. Any transient effects present in the ABL are expected to be generated by the turbulent inlet condition which is overlaid onto this mean flow.

3.7.3. Sinusoidal Turbulence

Sinusoidal turbulence can be defined in a number of ways and can become arbitrarily complex. The turbulence pattern created for this project involved the creation of two discrete sinusoidal surfaces which are overlaid onto one-another and normalised. The two surfaces move in different directions with each time step and have different magnitudes of contribution to the turbulent profile. While this approach only uses two sinusoidal surfaces, the approach could be extended to use many more for less predictable turbulence patterns. The only limitation of this method is the restriction of defining inlet turbulence in a singular direction to maintain continuity since vertical turbulence will cross shear layers and become unphysical. By only specifying flow-wise turbulent velocity the inlet relies on the model to generate full turbulence from evaluation of the Reynolds Stress tensor during simulation.

Since the turbulent profile is to be overlaid onto the mean velocity profile of the ABL, the magnitude of the sinusoidal turbulence should take into account the magnitude of the mean flow. Therefore, the generalised expression of sinusoidal turbulence is shown in equation 11 below.

Equation 11: $u = \bar{u}(z) \times \big(1 + f(z, y, t)\big)$

The basis of the turbulent function used is a sinusoidal function of the form shown in equation 12. This approach specifies a spatial frequency f_s, an amplitude A and two discrete phases p_y and p_z. The most important quantity in this equation is the spatial frequency since it can be equated to a turbulent length scale. This should be matched to the length scale expected in highest quantity in the flow as derived by experimental or arithmetic methods.

Equation 12: $f(z,y) = A \times \left(sin(f_s z + p_z) \times sin\left(f_s y + p_y\right)\right)$

In order to define the effect of time on the sinusoids, a secondary time-varying phase is added with a corresponding time frequency f_t. By changing the phase, the pattern will translate across the domain at an angle defined by the phase magnitudes as shown in equation 13.

Equation 13: $f(z,y,t) = A \times \left(sin(f_s z + p_z + p_{zt} f_t t) \times sin(f_s + p_y + p_{yt} f_t t)\right)$

$$where\ p_{zt} = \cos(\theta)\ \&\ p_{yt} = \sin(\theta)$$

This pattern on its own does not represent turbulence particularly well since is a repeating pattern moving in an easily predictable way. In order to create a more turbulent looking velocity field, several such patterns with discrete characteristics will be overlaid and normalised as shown in equation 14.

Equation 14: $f(z,y,t) = \sum_{k=1}^{N} \frac{A^k}{B} \times \sin\left(f_s^k z + p_z^k + p_{zt}^k f_t^k t\right) \times \sin\left(f_s^k + p_y^k + p_{yt}^k f_t^k t\right)$

$$where\ B = \sum_{k=1}^{N} A^k$$

The above equation represents the base implementation of a sinusoidal turbulent inlet and can be improved greatly with further work. However, this implementation is representative a relatively simple sinusoidal inlet and will be used during validation cases.

3.7.4. Validation

Based on the principals of turbulence in fluid simulation, a number of physical restrictions can be used to validate the model. The conservation of mass (or volume in incompressible flows) can be evaluated to ensure no numerical error is introduced from essentially adding or removing fluid to the flow during the calculation of turbulence. It is required of a turbulent inlet to leave the mean flow unaffected when the simulation is time averaged since any divergence would indicate the turbulence model is changing the mean flow from a physically correct solution. Both of these validation methods are concerned with mean values of the inlet velocity with respect to time.

The purpose of an inlet condition is to minimise the amount of space required for the numerically correct profile to be obtained. The validation of an inlet condition is not concerned with how accurately the flow profile is modelled but how quickly an accurate profile can be derived downstream. In order to examine the quality of this method of generating inlet turbulence it will be compared against other models including random noise, sinusoidal turbulence and uniform flow as a control. By examining the distribution of enstrophy within the channel, the development of the turbulent flow profile can be evaluated.

Equation 15: $\varepsilon(\boldsymbol{\omega}) \equiv \frac{1}{2}\int_S (\nabla \times u)^2 dS$

Enstrophy is a scalar calculated from the curl of the velocity as shown in equation 15. The magnitude indicates the degree to which finite volumes next to one-another vary in velocity. While this value cannot be used to quantify turbulence it is useful to qualitatively analyse the distribution of turbulence as well as show the shape of turbulent structures. The ideal turbulent inlet should provide a turbulence pattern which persists in the flow while being physical in its structure.

In order to validate the inlet conditions, they are simulated in an identical domain for a direct comparison. The domain in question is a long cuboid representative of flow between two parallel plates at a Reynolds number of 330,000. The top and bottom faces are defined as static walls and the side faces are defined as cyclic boundaries. The mesh is refined sufficiently that the y+ values throughout the domain are below 1 as per the LES conditions. The internal field will be initialised with mean flow values as determined by a RANS simulation with the turbulence fed in at the inlet. The simulation will be run sufficiently long for the turbulent flow to reach the end of the domain and allow the full formation of boundary layers within the domain.

The validation process will be completed for three different inlet conditions: synthetic eddies, sinusoidal and random noise (provided by OpenFOAM's *turbulentInlet* boundary condition). Readings of the flow profile will be taken at 1m intervals along the length of domain on the central plane. This will be the basis of evaluating the accuracy of the inlet condition to stable turbulence. Also, the velocity will be read on the inlet face to check for continuity of the various methods.

4. Presentation of experimental or analytical results

The results presented in this section relate to several different runs on the same test domain using various inlets. By evaluating these results, the synthetic eddy method inlet can be verified as accurate and compared against other methods for an indication inlet quality.

The test domain used for the evaluation of the various inlets is representative of flow between two parallel plates. As flow passes the plates, a boundary layer will form and eventually reach a constant size. This holds true regardless of the inlet since the wall effects will cause shear layers which will provoke turbulent behaviour in otherwise laminar flow. As such, the quality of an inlet can be directly correlated to the distance from the inlet at which the boundary layer stabilises to its final characteristics.

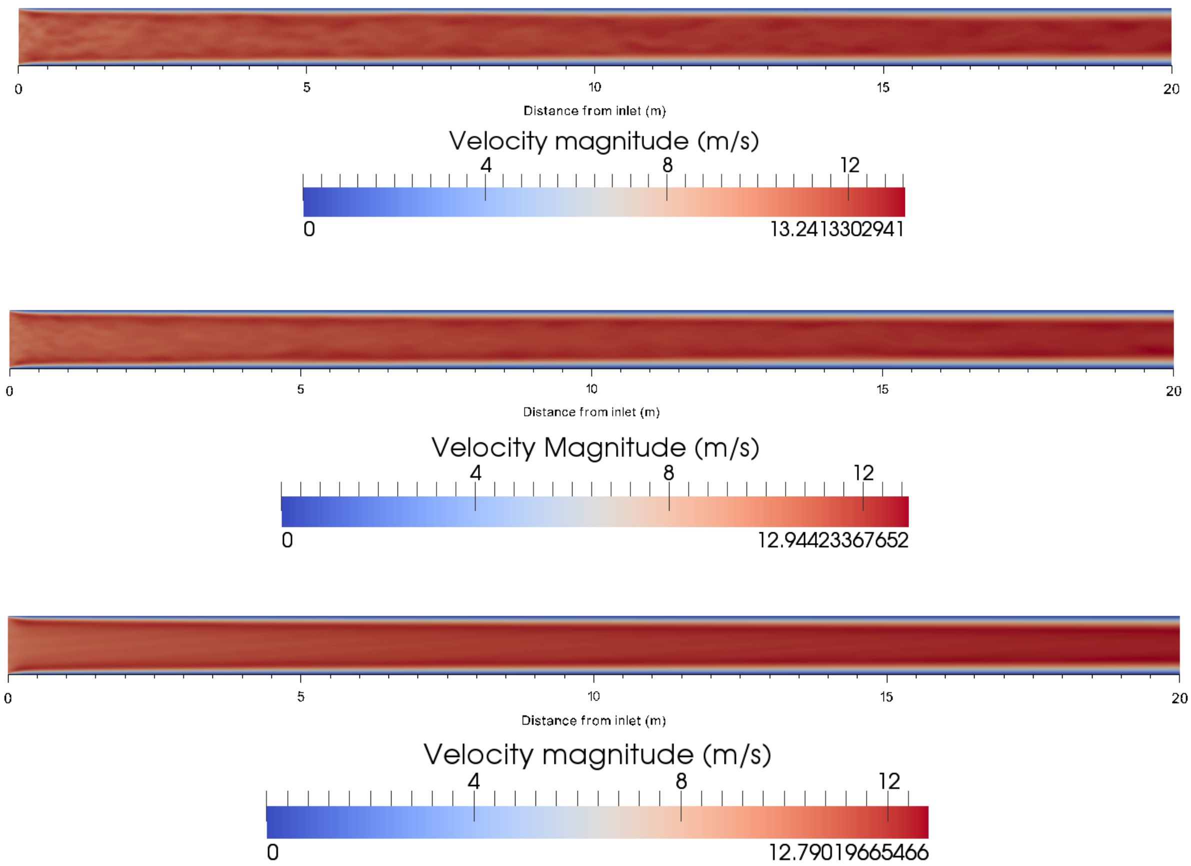

Figure 1: Cross section of test domain for various inlet conditions: 1-SEM, 2-random noise, 3-sinusoidal

Figure 1 shows a plot of instantaneous velocity magnitude through a cross-section of the domain. Since it is known that the flow in this domain is turbulent, some turbulence is expected to be present within the domain during the simulation. The turbulence at the inlet is unlikely to mimic the actual turbulent structures in the flow exactly so the turbulent profile is expected to develop throughout the domain to reach a steady turbulent state. The quality of the inlet can be judged by the manner that turbulence propagates within the domain.

The profile of the simulations with SEM and random noise inlets appear quite similar close to the inlet. Both show random distributions with local minima and maxima. The magnitude of the minima and maxima are more pronounced in the SEM inlet model which is indicative of the presence of a length scale. This is a benefit over a cell-based random method since the length scale for turbulence using such an approach is directly related to the grid scale. Further into the domain the turbulence associated with the random noise inlet becomes more blended as the solver attempts to average out cells with large velocity gradients to its neighbouring cells. This blending can be attributing to the lack of coherent structures present in the SEM and sinusoidal inlets to preserve the turbulence. The turbulent pattern in the sinusoidal inlet is a repeating pattern which changes its phase with time. The pattern can be seen prominently at the inlet and can be seen to persist to the outlet. This turbulence pattern is clearly unphysical and is representative of how turbulent structures will persist in the flow in LES models. This is the justification of creating turbulent structures at the inlet which are physical in nature. Were a perturbation on the flow introduced the turbulent behaviour would propagate throughout the domain. However, a procedure such as this is unreliable and computationally costly.

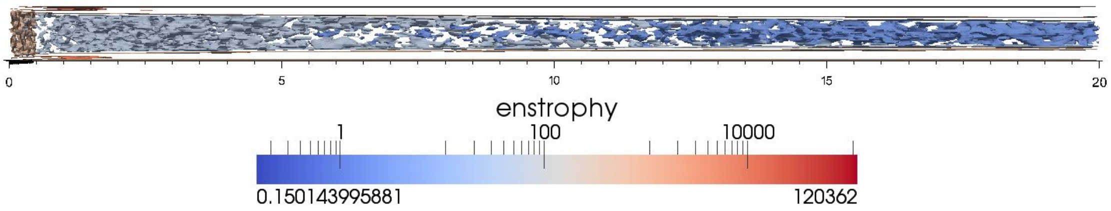

Figure 2: Enstrophy iso-surfaces for test domain with SEM inlet

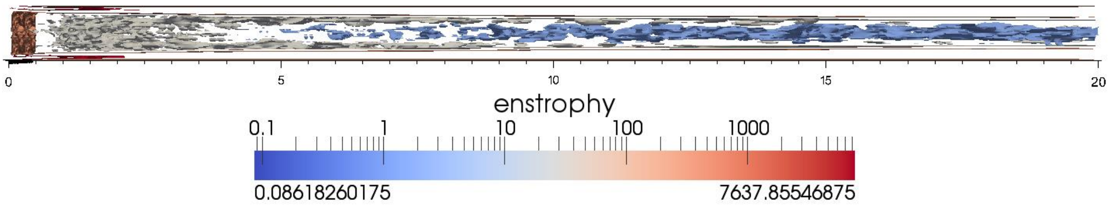

Figure 3: Enstrophy iso-surfaces for test domain with random noise inlet

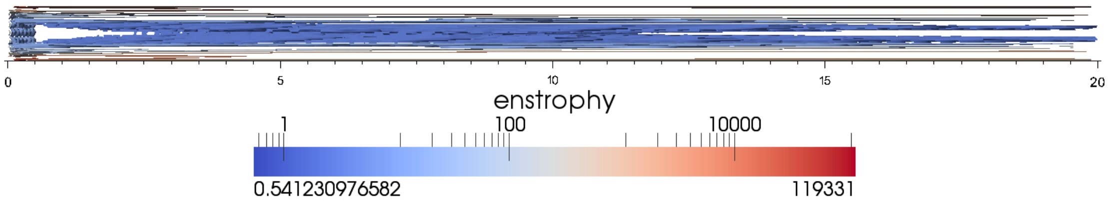

Figure 4: Enstrophy iso-surfaces for test domain with sinusoidal inlet

The enstrophy iso-surfaces shown in the figures above are representative of the distribution of turbulence throughout the domain. It can be seen that the three inlet conditions present different behaviour in terms of enstrophy throughout the domain due to their respective characteristics. The domain with the SEM inlet shown in Figure 2 displays an even distribution of turbulence across its inlet face. The enstrophy surfaces indicate the presence of the eddies throughout the domain, marking a loss in turbulent energy as the contour gradually changes between 5m and 10m from the inlet. The domain with the random noise inlet shown in Figure 3 displays a more noisy distribution of enstrophy by the inlet which decays away at around 4m into the domain allowing a more natural turbulence pattern to be generated by the model. At the outlet this new turbulence dominates the free stream while the boundary layer still retains a degree of turbulence from the inlet. This divergence between the inlet turbulence and model turbulence is a sign of a poorly defined inlet and is not present on the SEM equivalent shown in Figure 2. The domain with the sinusoidal inlet shown in Figure 4 shows the sinusoidal turbulent structure persists to the outlet and appears to take much longer to dissipate. This shows how an incorrect inlet specification can cause the failure of a model even significantly downstream of the inlet patch.

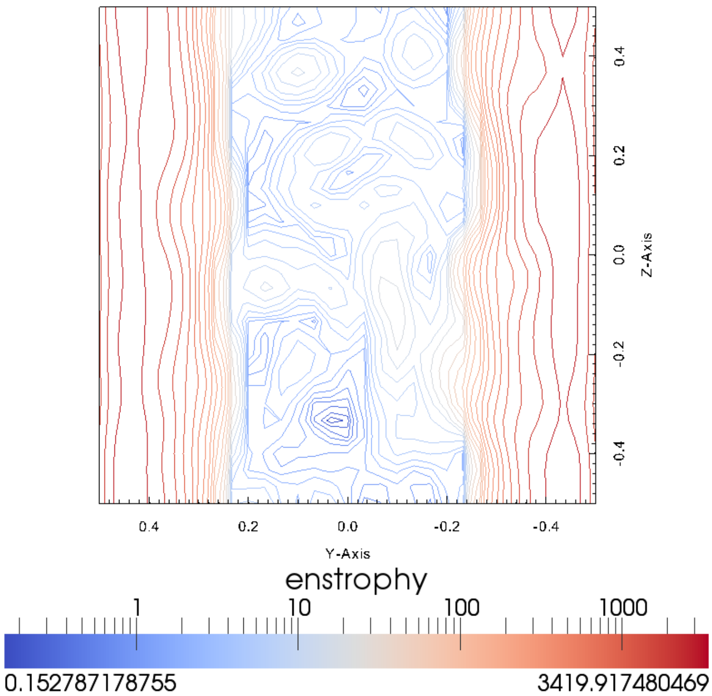

Figure 5: Contours of enstrophy on domain outlet using SEM inlet condition

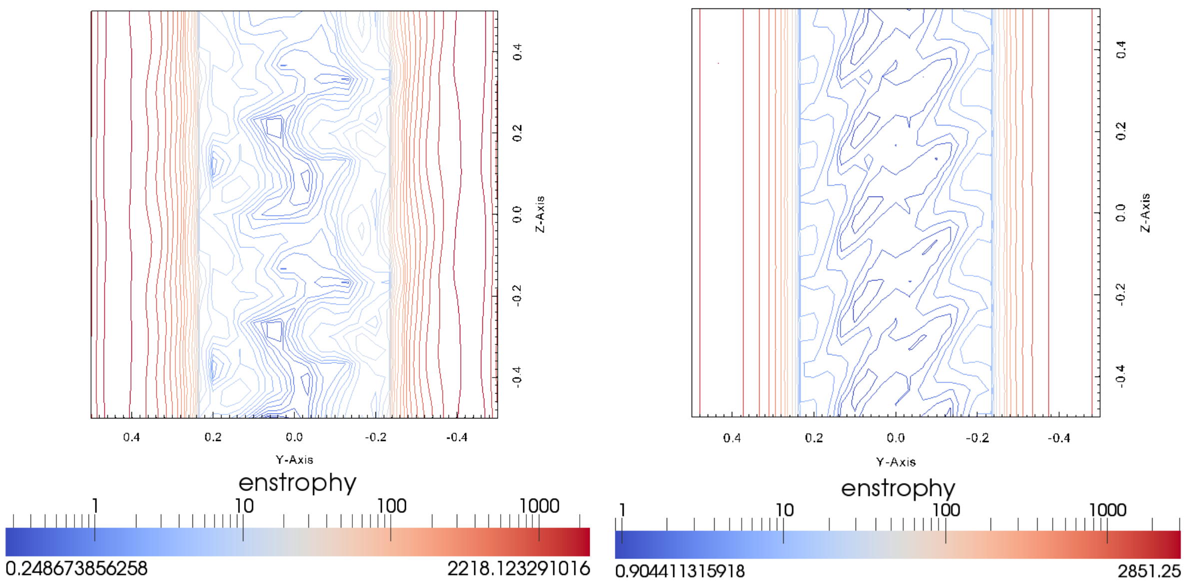

Figure 6: Contours of enstrophy on domain outlet using random noise (left) and sinusoidal (right) inlet conditions

The figures above show the enstrophy distribution on the outlet faces of the respective domains. A comparison of these shows the relative strengths and weaknesses of the various inlets at defining turbulence in the downstream. The sinusoidal structure is still clearly visible even at the outlet of the test domain indicating a simplistic model of sinusoidal turbulence may hinder a simulation rather than help it. Comparison against a more complex sinusoidal inlet would be required to properly validate the SEM inlet against it. The random noise and SEM outlet contour look similar but it can be seen that the turbulence within the random noise domain is constrained to the free-stream whereas the turbulence in the SEM domain interacts with the boundary layer. The latter is a more physical response since turbulence is to be distributed throughout the domain.

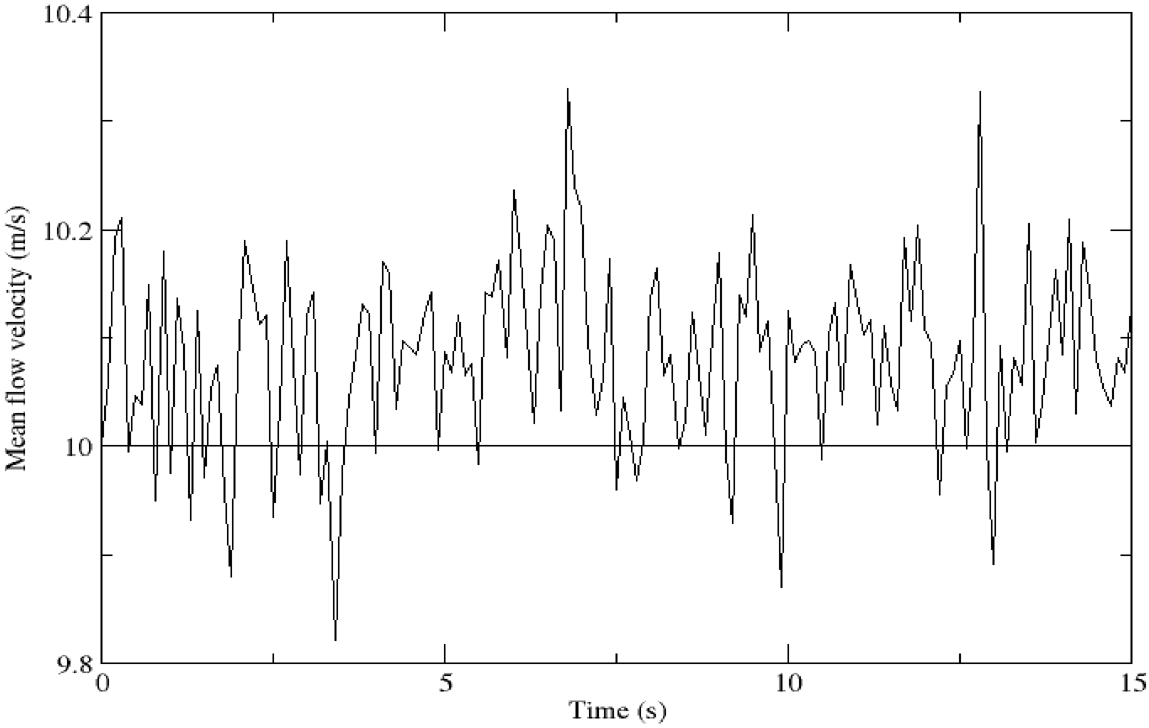

Figure 7: Graph of mean velocity of SEM inlet against time

Figure 7 shows a plot of the mean flow velocity averaged over the SEM inlet face. Fluctuations in the graph are representative of the random nature of the SEM inlet, because there is no guarantee that a given time step will have equal contribution from eddies with opposite intensities to balance the mean velocity. In order for continuity to hold, the average of these mean velocity should average out over a long time to the inlet mean velocity the eddies were originally created with. In this case, the mean velocity was specified as 10m/s and the results suggest an average mean velocity marginally higher than this.

This average value is expected to be close to the mean flow velocity before turbulence is added which in this case is 10m/s. The graph shows the mean velocity fluctuates around a mean value of 10.1m/s. On top of this, the magnitude of the fluctuations around this mean do not appear to be converging or diverging from it. The reason for this error is likely due to a combination of factors. The fact the average of the mean velocities over the face is marginally over the expected value could be attributed to averaging over too short a time or with too few values. It could also potentially be due to errors within the RANS precursor simulation due to the presence of wall functions at the point of sampling. Another possibility is due to the random nature of the eddies which is also the likely cause of the fluctuations around the mean values. This is the intended behaviour of the eddies being simulated since a random distribution presents a more physical result.

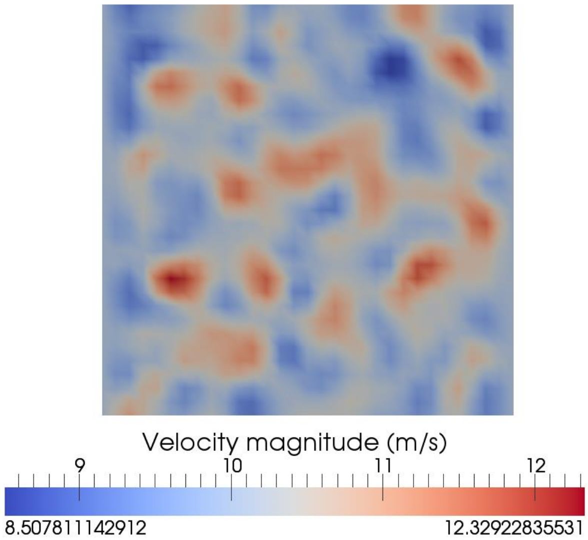

Figure 8: Plot of velocity magnitude for SEM inlet

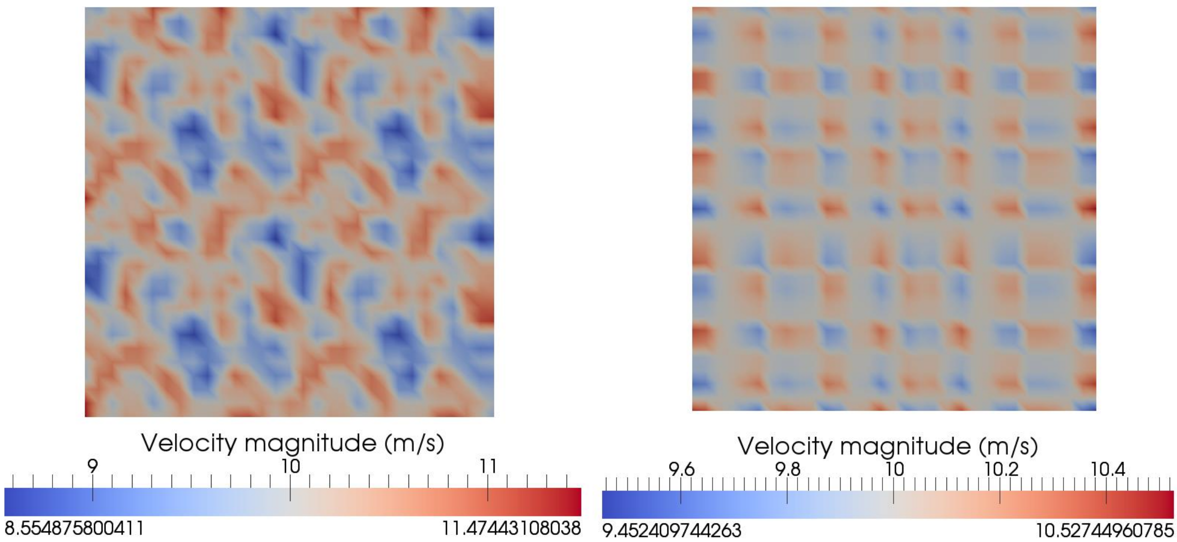

Figure 9: Plot of velocity magnitude for random noise (left) and sinusoidal (right) inlet conditions

Figure 9 shows the velocity profiles of the various inlet conditions on a single time step. These can be evaluated on their likeness to coherent turbulent structures. This a is a qualitative check with the aim of ensuring that any method of generation of inlet turbulence generates a physical looking result as opposed to a unphysical numerically correct one. The sinusoidal inlet can be seen to have a clear repeating pattern. This is a reflection on its definition as the superposition of two sinusoidal surfaces described earlier. By increasing the number of sinusoidal surfaces present, the interference pattern would become more complex presenting a less obvious repeating pattern. The frequency spectrum could be maintained by specifying multiple sinusoids of the same frequency with different phases. In order for the continuity constraint to be met by the sinusoidal inlet, the sinusoids must be specified such that the maximum wavelength is not significantly greater than inlet in size, increasing the likelihood of repetition in the inlet. The random noise inlet also shows a degree of repetition with is reflective on its implementation within OpenFOAM. This repetition is likely due to the superposition of the same random field superimposed at different scaling in a uniform manner. This ensures a degree of coherence at the cost of generating a repeating pattern. Unfortunately, the same coherence is not present temporally as a new random field is generated each time step. The SEM inlet does not show any repetition although it is random in nature. The structures visible on the SEM inlet face appear coherent to one another while maintaining distinct minima and maxima.

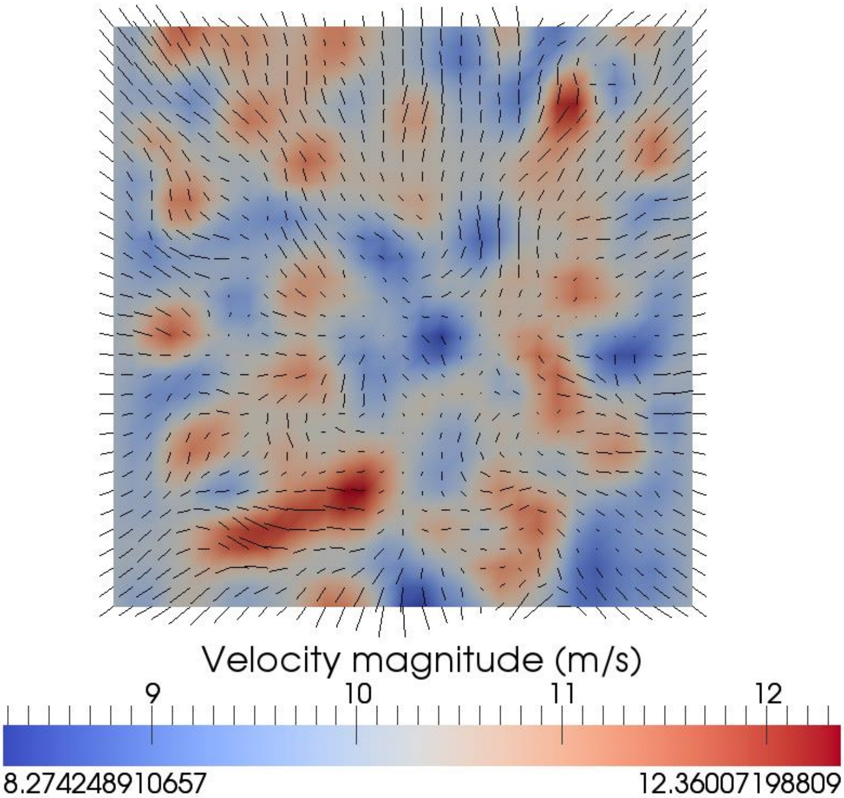

Figure 10: Velocity magnitude of SEM inlet at single time step showing swirl capability

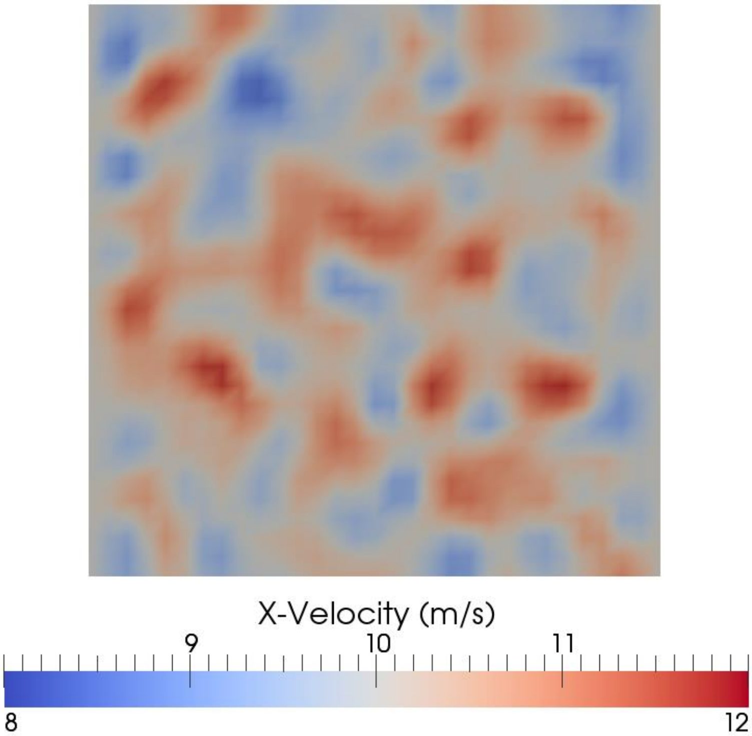

Figure 11: Stream-wise velocity of SEM inlet at single time step

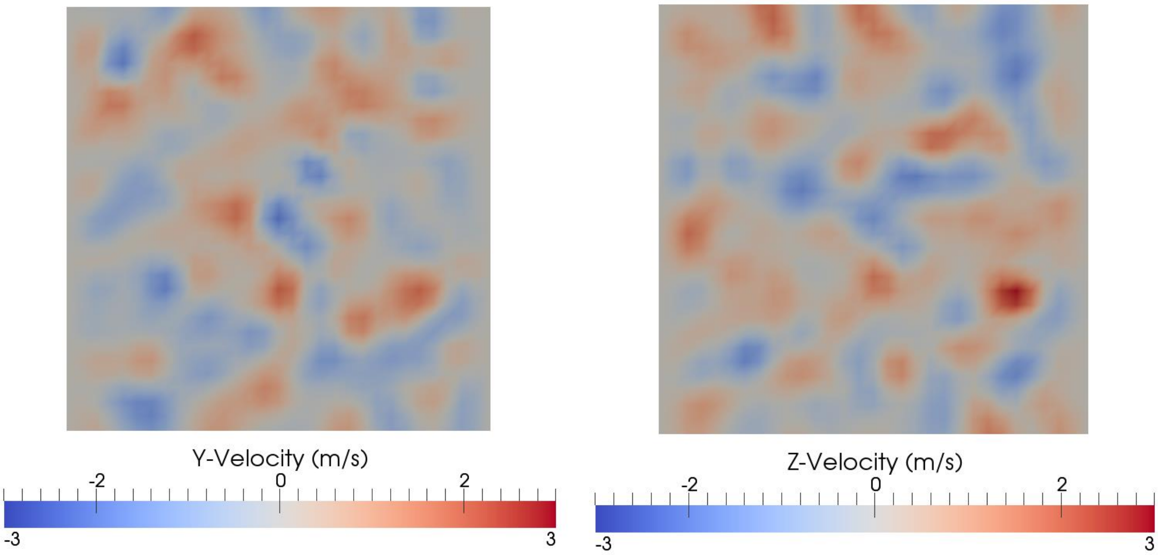

Figure 12: Flow velocity of SEM inlet perpendicular to free-stream direction at a single time step

The SEM inlet has the capability of producing a turbulence pattern contiguous in all directions as shown in Figure 10. A breakdown of the velocity profile in each of the axis directions are shown in Figure 11 and Figure 12. These give an indication of the manner in which eddies are distributed and their effect on the inlet's specification. By considering similarities between these plots it can be seen how individual eddies influence the flow respective of the component of the velocity. Toward the bottom left of these plots a 'hotspot' can be seen (most clear on z-velocity plot). This point represents the contribution of an eddy that has a positive intensity for all three directions since there is a visible positive magnitude for this point on all the plots. It is immediately clear that the distribution of velocity in each of these plots is random but contiguous to the values around it. Also, the values in each of the plots appear distributed around a mean value with no large divergences from this average value. These plots demonstrate the inlets capacity to qualitatively look like turbulence which is a factor which is a reasonable consideration when selecting an inlet method.

One of the factors important to the validation of a turbulent inlet is the qualitative physicality of the inlet. The generally accepted perception of turbulence is a superposition of coherent turbulent structures and any inlet condition should aim to replicate this. It is rather difficult to quantify the presence of coherent structures in the flow. Coherence can be considered in two ways: spatial coherence and temporal coherence. If cells in the inlet plane have velocity values significantly different from one another such as in a random noise inlet, there is no apparent coherence in the inlet and the noise will be filtered out by the model to a more coherent turbulence pattern derived from the noise. Also, is the velocity of an inlet cell changes dramatically between time steps, the difference will be averaged out when correcting for flow continuity. Figure 10 shows the SEM inlet plane at a given time step and it can be seen that the inlet displays spatial coherence in all three directions as the pattern appears smooth in terms of velocity magnitude and direction. Also, due to the nature of the eddies within the program, if the length scale and time step are specified correctly the same eddy will affect any given inlet cell on a series of time steps offering a degree of coherence to the temporal characteristics.

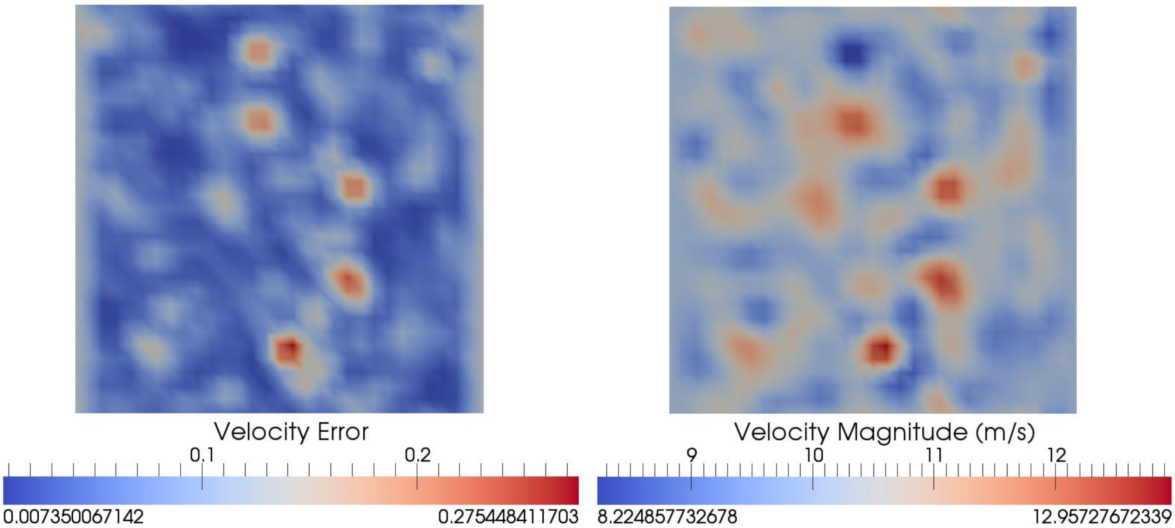

Figure 13: Plot of error present in SEM inlet (values between 0 and 1) and corresponding velocity magnitude for time step

It was found during this project that simulation of the eddy field upon itself would cause a compounding of errors. These errors were shown to not be self-stabilising since the model was unable to return the flow characteristics to their previous magnitudes. The cause of this error was inherent in the approach since small numerical errors are present in the simulation. When used to calculate the altered velocity field, the aim is to change the direction of the velocity vector based on the contribution of eddies while maintaining the magnitude. Figure 11 shows a plot of error which is 0 where the magnitude of the altered field is the same as the initialising field and a value of 1 where the magnitude diverges from the original by 100%. The largest errors occur on hotspots which indicate the presence of a number of eddies affecting the same cell with the same intensities. These errors are not linked to the number of eddies since they are still present when only a single eddy is simulated within the flow. The most likely cause is an inaccuracy in the Reynolds stress tensor generated by the model. This theory is backed up by the higher error values close to the wall where an accurate Reynolds stress value is harder to interpret. Simulation of the SEM method on a significantly more refined domain would be required to verify the cause of the error. However, by taking the approach of decoupling the eddy velocity field from the velocity field, the error values can be kept constant to be averaged out by the model. This small degree of error could be the cause of the small divergence from expected mean flow velocity shown in Figure 7.

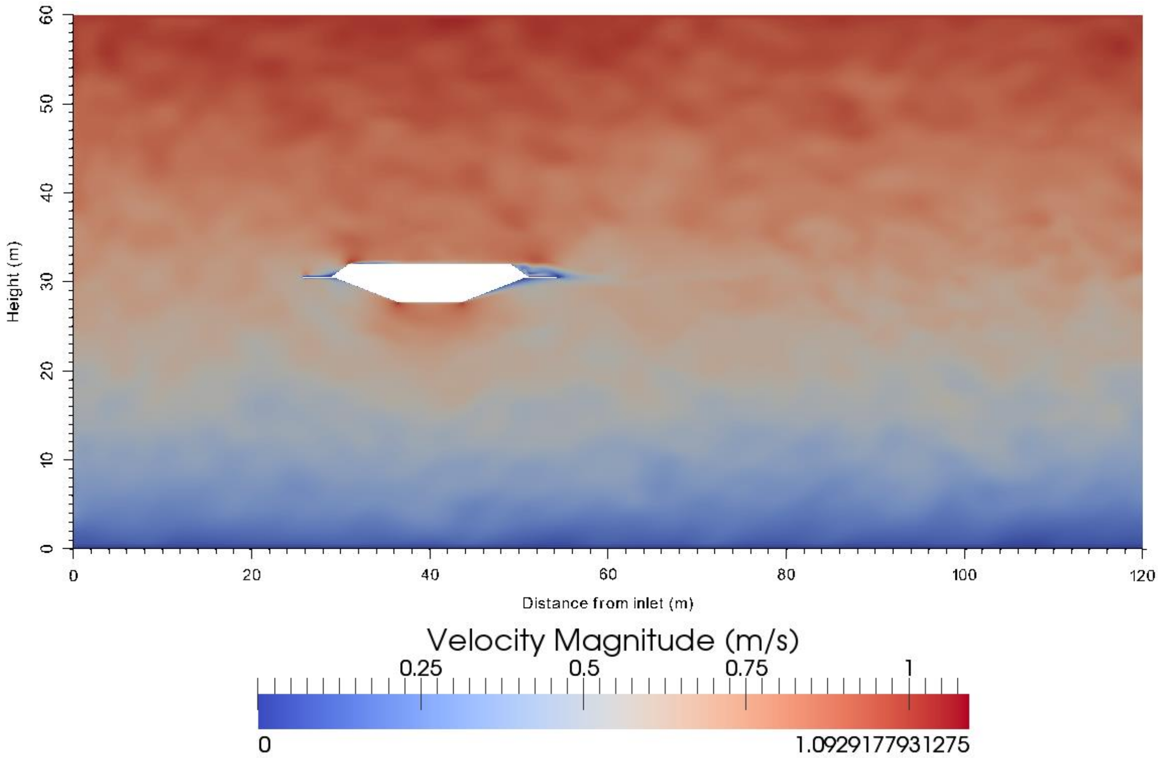

Figure 14: Synthetic Eddy Method inlet simulating atmospheric air flow past a bridge deck (group project simulation)

The aim of this project was to create an inlet capable of accurately simulating atmospheric air flow for use within the group project. Figure 14 shows a velocity plot of a simulation set up as part of the group project. While this simulation is the result of a collaboration of three people the inlet condition and field initialisation was done as part of this individual project. The mean atmospheric velocity profile was gathered from another group member and used to initialise the field of the test domain and used as the inlet for the precursor simulation. The SEM solver was used on the precursor simulation to generate inlet data as well as used for a single time step on the test domain to initialise the field with turbulence. This approach saves a significant amount of time in the simulation since physical turbulence is already present within the domain. Due to the scale of the simulation required to model atmospheric flow a full validation and analysis of this simulation was not possible within the time scale of the project.

5. Discussion and conclusions

A method of generation of inlet conditions based on the simulation of synthetic eddies on top of a precursor RANS simulation has been created to properly define the turbulent velocity profile. This method is an adaption of an existing method made for use with OpenFOAM which had no publically available implementation prior to this project. Development of this method gained insight into the practical issues of such an approach to inlet specification. Numerical accuracy is inherent in the approach due to the discretisation of the problem limiting the method to acting as an interface between a decoupled test case and precursor simulation. This approach prevents the compounding of errors but places a higher computational cost on the generation of inlet conditions. Additionally, the reliance on the accuracy of the Reynolds stress tensor places high significance on the accuracy of the turbulence model with dramatically different results generated from different turbulence models for the same flow conditions.

The synthetic eddy method described is shown to be as numerically accurate under limited test conditions as comparable inlet conditions, namely random noise and basic sinusoidal turbulence. The benefit of the synthetic eddy method is likely more evident in more complex flows with a non-uniform distribution of Reynolds stress across the inlet face. Additionally, the coherence of the structures present in an SEM inlet allows a longer persistence of turbulence within the flow presenting a more accurate turbulent profile at the test surface rather than at the inlet alone. This coherence is displayed in plots of enstrophy showing the presence of turbulence with an even distribution throughout the domain. The group simulation using the inlet condition was able to run successfully and remained stable and physical throughout. Based on this simulation the inlet was shown to be capable of representing these large scale flows such as those present in the atmospheric boundary layer. Further analysis of cases simulating the atmospheric boundary layer would be required to validate the inlet's use in these conditions. However, due to the high computational cost associated with simulating atmospheric flow this was not possible within the scope of this project.

A more rigorous validation of this method would involve the simulation of a domain with the inlet condition with a more refined grid scale. This would allow for the eddy characteristic length to approach the true value since it is limited by the grid scale in the experiments described in this paper. The refinement of the test domain would also require smaller time steps to be taken to maintain temporal numerical accuracy prescribed by the courant number. The duration and scale of such a simulation make it unsuitable for the scope of this project since it would take many weeks to complete on available equipment.

Further work on the implementation of the inlet condition itself could aid in the accuracy of the inlet condition as well as provide some improvements to its utility. The implementation could be adapted to operate on top of a time-invariant velocity profile present at the inlet of the test domain removing the need for a precursor simulation. Alternatively, the method could be adapted to run from previous solutions by storing the properties of the eddies within the flow at given time steps. This would allow for inlet conditions to be extended only when required and improve the practicality off the inlet generation approach.

The implementation described above treats the eddies within the flow as independent from the mesh. They are transported by a mean flow, allowed to exist outside of the mesh while influencing cells within it and retain a fixed length scale. Further development of the method may involve the management of the eddy length scale with position and time, movement of the eddies by localised flow and the definition of eddy behaviour as they pass cyclic, symmetric and wall boundaries. Previous papers have also shown that the shape function can be changed to create a more physical turbulence pattern. Deriving a method by which the shape function could be based on the frequency spectra at locations in the mesh would likely produce the most accurate result.

6. Project management, consideration of sustainability and health and safety

6.1. Project Management

Project management ensured that goals were met while keeping control of deadlines and maintaining awareness of the greater group project’s progress. The group project was a research project and as such no clear conclusions could be drawn about how long aspects of the project would take and even if they would yield results. Throughout this project the initial plans have had to be reviewed and methods adjusted to yield for inaccurate time frames.

Before the completion of the SEM inlet, time was allocated to the creation of a lesser sinusoidal inlet to allow the simulation of the group model with a reasonable turbulence structure. Such a decision was reached by careful management of the project as an individual and as part of a group. The methods of management used are further explained below.

The main approach managing time was attending a weekly group meeting. This allowed the progress of the individual project and the group project to be evaluated on a comparable scale so the rate of work completion can easily be seen. Also, by setting attainable goals to be completed by the next week's meeting progress could be guaranteed. On top of this, knowing what other members of the group were doing allowed targets to be tailored to best help them and by effect the group project as a whole.

A method employed to manage the project on an individual basis was the use of a logbook. By compiling all the notes made on the project within a logbook, less time is wasted researching information already found. Since the logbook is kept for personal use only, it can be used on an ad-hoc basis to suit the needs at the time in the project.

The methods of time management mentioned above indicate the management of time on a small scale to ensure the steady progression of the project. In order to evaluate the progression of the project on a larger time scale, a Gantt chart was created prior to the projects commencement. A Gantt chart is a useful tool since it shows how the project as a whole is broken down into a number of tasks and how these tasks influence one-another. At any given point in time, the Gantt chart can be reviewed to see what task should be worked on, how long until it needs to be completed, what tasks are required to begin it and if the task is running ahead of or behind schedule.

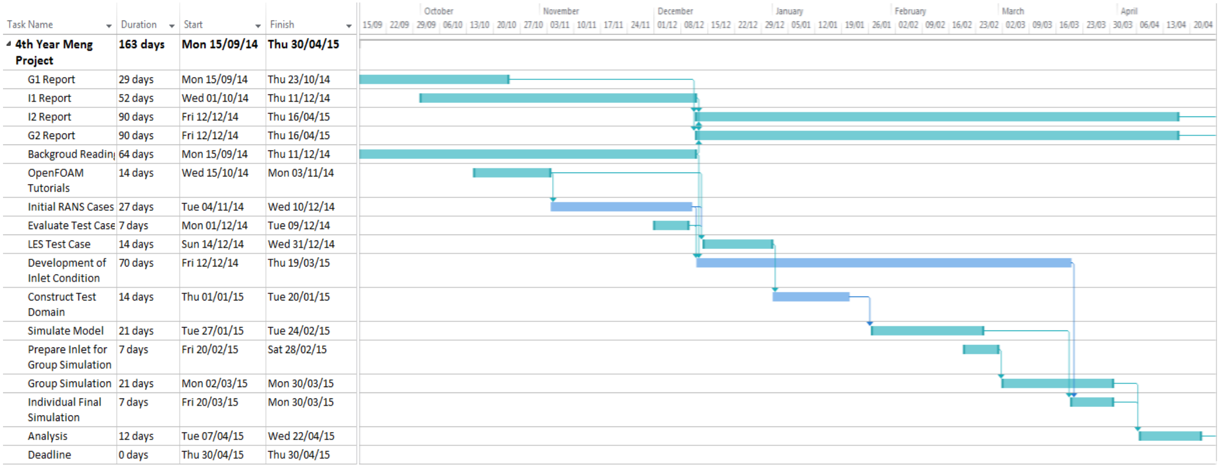

Figure 15: Gantt chart for project

As shown by the Gantt chart in Figure 15, a large part of the project is dedicated to the creation of the inlet condition. It is difficult to manage this task since it cannot be guaranteed to progress with time. Throughout the process of implementing the synthetic eddy model errors were encountered with no immediate solution. Finding the solution for these errors required careful thought about the operation of the program and the fluid mechanics it is simulating. Only after progressively observing, evaluating and fixing errors within the program over several weeks did it reach a stage where it was stable enough to run simulations. Additionally, the computational cost associated with running a simulation was taken into account when managing the time in the project. Time was allocated to allow simulations to run for several days and in the case of the group simulation, several weeks. By collaborating with the group it was ensured that there was adequate time to run simulations, account for any problems and analyse data within the allowed timeframes.

6.2. Health and Safety

The undertaking of this project involved a significant amount time spent working with computers. During this project entire days have been spent setting up various cases and creating the inlet condition. As such, several hours a day of looking at computer screens with minimal physical exercise pose a potential risk to health which requires addressing. A risk assessment was performed to better understand the hazards associated with working in such conditions which is summarised below.

The practical hazards of a situation such as this encompass a number of areas. Eye strain can occur from looking at a screen for too long consecutively as well as looking at a screen in unfit conditions (dark room, glare on the screen, etc.). Additionally, a deterioration of physical health can occur from sitting in front of a computer for long stretches of time on a regular basis. Typing for long periods of time, a hazard particularly relevant to programming and scripting on a heavily text-based program such as OpenFOAM can lead to repetitive strain injuries (RSI) and can lead to long term joint damage. Also, the equipment used in this project is liable to fail and can pose a risk to safety is poorly maintained.

Reduction of the risks associated with the project is best approached in a preventative manner. When beginning work on a computer the seat and screen should be adjusted for a good working posture and view with minimal strain to the eyes. Periodically, breaks should be taken to walk around away from screens to reduce the risk of damage to eyes. Should any significant discomfort be experienced during the project as a result of working conditions, the safety procedure should be reviewed and amended as necessary along with appropriate actions being taken to address the concern at the time. The workspace should be kept clean and clear of hazards such as loose cables and large objects which can be tripped over. Equipment should be maintained regularly and checked for faults before use. By observing these precautions, reacting to any accidents or dangerous situations and reviewing the procedure when required, a safe working environment can be ensured throughout the project.

7. Contribution to group functioning

The aims and objectives detailed in this report are designed to coincide with the goals of the larger group project. The group project in question involves the study of aero-elastic effects on bridge deck structures. In order to best evaluate the problem the group is subdivided into two sub-groups focussed around structural analysis and computational fluid analysis with this report contributing primarily to the latter.

The goal of the fluids sub-group is to create an LES based simulation of atmospheric air flow past a bride deck using OpenFOAM. A key part of that goal involves defining an inlet condition capable of replicating both the mean atmospheric flow profile and the turbulent flow profile. Without a correct inlet the simulation would either fail due to incorrectly defined flow or become significantly more computationally costly due to the reliance on an inlet region to allow flow to develop within the model.

For the purpose of creating this inlet condition, a basic LES model was set up which was later refined by another group member, Joe, to be more suitable for the final simulation. Another group member, Josh, was focussed on analysing the effects of terrain on the mean air flow and contributed a mean atmospheric airflow profile. Prior to the completion of the synthetic eddy inlet condition, a sinusoidal inlet condition was created using the mean atmospheric air profile to allow some results to be generated. The final mesh of the bridge deck represented a collaboration of all members of the fluids subgroup simultaneously developing the mesh to generate the best one possible using the bridge model created by a member of the structural subgroup, Damien.

Beyond this, a later simulation was run using a more complex inlet condition overlaying synthetic eddies onto a mean air profile derived by Josh. The inlet condition used for this simulation represents the objective of this report since it accurately simulates a turbulent time variant atmospheric profile that is physically representative of real air flow.

The results derived from this case and the prior case using a sinusoidal turbulence model was used to find information about the forces caused by the vortex shedding in terms of position, amplitude and frequency. These were handed over to the structural subgroup for comparison against other evaluation methods and evaluation of the bridges dynamics when subjected to such forces.

Due to the focus of this individual project on the specification of inlet conditions, a degree of separation between the aims of this project and that of the structural subgroup is present. By undertaking a project heavily focused in fluid simulation the results generated by the fluids subgroup are of a higher quality than they would be otherwise thereby helping reach the goals set by the overarching group project.

8. References

[1] C. Moeng and P. Sullivan, “Large Eddy Simulation,” in *Encyclopedia of Atmospheric Sciences*, 2002, pp. 1140-1150.

[2] M. Charles and P. Sagaut, in *Large eddy simulation for incompressible flows: an introduction*, Berlin, Springer Science & Buisness Media, 2006, p. 4.

[3] G. Tabor and M. Baba-Ahmadi, “Inlet conditions for large eddy simulation: A review,” *Computers & Fluids,* vol. 39, no. 4, pp. 553-567, 2010.

[4] L. Davidson, “Hybrid LES-RANS: Inlet Boundary Conditions for Flows with Recirculation,” *Advances in Hybrid RANS-LES Modelling,* vol. 97, pp. 55-66, 2008.

[5] I. Veloudis, Z. Yang and J. McGuirk, “Novel Implementation of a Digital Filter Based Approach for the Generation of LES Inlet Conditions,” *Flow Turbulence Combust,* vol. 79, pp. 1-24, 2007.

[6] H. Lim, T. Thomas and I. Castro, “Flow around a cube in a turbulent boundary layer: LES and experiement,” *Journal of Wind Engineering and Industrial Aerodynamics,* vol. 97, no. 2, pp. 96-109, 2009.

[7] T. Andrianne, M. A. Razak and G. Dimitriadis, “Flow Visualisation and Proper Orthogonal Decomposition of Aeroelastic Phenomena,” Belgium, 2011.

[8] N. Jarrin, S. Benhamadouche, D. Lawrence and R. Prosser, “A Synthetic-Eddy Method for Generating Inflow Conditions for LES,” *International Journal of Heat and Fluid Flow,* vol. 27, pp. 585-593, 2006.

[9] N. Jarrin, A. Revell, R. Prosser and D. Lawrence, “Reconstruction of Turbulent Fluctuations for Hybrid RANS/LES Simulation Using a Synthetic-Eddy Method,” in *7th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements (ETMM7),* Limassol, Cyprus, 4-6 June, 2008.

[10] N. Jarrin, “Synthetic Inflow Boundary Conditions for the Numerical Simulation of Turbulence,” 2008.

[11] D. Muñoz-Esparza, B. Kosović, J. van Beeck and J. Mirocha, “A scoastic pertubation method to generate inflow turbulence in large-eddy simulation models: Application to neytrally stratified atmospheric boundary layers,” *Physics of Fluids,* vol. 27, p. 035102, 2015.

[12] H. Jasak, A. Jencov and Z. Tukovic, “OpenFOAM: A C++ library for complex physics simulation,” 2013.

[13] swak4Foam, “swak4Foam,” 2014. [Online]. Available: https://openfoamwiki.net/index.php/Contrib/swak4Foam. [Accessed April 2015].

[14] J. Garratt, The Atmospheric Boundary Layer, Cambridge: Cambridge University Press, 1992.

[15] P. Mason, “Large-Eddy Simultion of the Convective Atmospheric Boundary Layer,” *Journal of Atmospheric Sciences,* vol. 46, pp. 1492-1516, 1989.

[16] L. Lignarolo, C. Gorle, A. Parente and C. Benocci, “Large eddy simulation of the atmospheric boundary layer using OpenFOAM,” in *13th International Conference on Wind Engineering,* Amsterdam, The Netherlands, 2011.

[17] A. Kolmogorov, “A refinement of previos hypothesis concerning the local structure of turbulence in viscous incompressible fluid at high Reynolds number,” *Journal of Fluid Mechanics,* vol. 13, no. 1, pp. 82-85, 1962.

[18] Y. Chang and A. Scotti, “Modelling unsteady turbulent flows over ripples: Reynolds'averaged Navier -Stokes equations (RANS) versus large-eddy simulation (LES),” *Journal of Geophysical Research: Oceans (1978-2012),* vol. 109, p. C09012, 2004.