5-2019

Turbulent Effects on Building Pressure using a Two-Dimensional Finite Element Program

Jonese Woods
University of Arkansas, Fayetteville

TURBULENT EFFECTS ON BUILDING PRESSURE USING A TWO- DIMENSIONAL FINITE ELEMENT PROGRAM

Jonese Woods

May 2019

Department of Civil Engineering
College of Engineering
University of Arkansas

ACKNOWLEDGEMENTS

I would like to sincerely thank my thesis advisor, Dr. R. Selvam for his guidance and support throughout the entire process of completing this project. Dr. R. Selvam served as a mentor for not only this project but he also offered valuable life lessons that would help me to excel in all facets of life. I am also thankful for the opportunity to use Dr. Selvam's finite element program and Tecplot software.

My sincere thanks are extended to my honor committee members, Dr. R. Williams, and Dr. G. Prinz. Thank you for the time spent reviewing this thesis.

I would also like to acknowledge the partial support received from the National Science Foundation under award number CMMI-1762999.

TABLE OF CONTENTS

ABSTRACT

Much of the knowledge about building aerodynamics today was obtained from physical testing like wind tunnel testing. Physical testing is time-consuming and very expensive. As a result, computational methods like the finite element method are being explored for use in building aerodynamics. Despite several years of research, there is still not a clear understanding of the peak pressure on buildings due to turbulence. Many of the research thus far has focused on comparing different computational methods. However, much work is needed in understanding the capability of the individual methods. In this work, a two-dimensional finite element program is used to investigate the effects of turbulence on building pressure. The flow around a square was investigated and the pressure at four different locations was monitored. A single sine wave was introduced at the inlet to simulate turbulence within the model. Changes were made to the amplitude of the wave and the wavelength and the variation in pressure was monitored. A comparative study of flow with turbulence and without turbulence was also conducted. Results have shown that in the model without turbulence, there is still pressure variation and the sinusoidal properties of the wave were still developed due to building effects. It was also found that waves of a smaller wavelength and amplitude had more pressure variation and a higher peak pressure. A wavelength less than or equal to the building height is considered a small wavelength. The formation and transport of vortices also influences the peak pressures observed. From the results obtained in this study, it was concluded that the pressure variation within the model was due to a combination of the sinusoidal properties of the wave and the formation and transport of vortices around the building.

CHAPTER 1: INTRODUCTION

1.1 General Overview

Wind Disasters like storms, hurricanes, and tornadoes are responsible for tremendous damage to infrastructure, injury, loss of life and economic damage. Since the early 1980's the intensity, duration and frequency of stronger storms have increased and are expected to continue increasing in years to come. Therefore, a greater understanding of the wind loadings on buildings is needed.

Despite several years of research, there is still not a clear understanding of the peak pressure on buildings due to turbulence. Hence, infrastructure may not be adequately designed to withstand strong wind forces. To understand the effects of turbulence on building pressure, researchers employed the use of physical test methods like wind tunnel experiments and field testing. Wind tunnel tests and field testing are expensive and very time-consuming. Another limitation to the use of wind tunnel testing in analyzing wind-building interactions is that "low-frequency effects of turbulence are damped due to scale limitations", Selvam et. al (2008).

Throughout the past 30 years, extensive research has been conducted to understand the peak pressure on buildings and the effects of turbulence on building pressure. Turbulences causes velocity and pressure to fluctuate in very short periods of time. Many of the work researchers have done for wind engineering focuses on the comparison of different types of turbulence models, while not many studies pay attention to the influence of the turbulence parameters in numerical investigations. (Wei Yang and Yong Quan, 2008).

With advances in computer technology, Computational Fluid Dynamic (CFD) is now a prominent tool used in wind engineering. Computational models are more efficient and can analyze greater amounts of data in a shorter amount of time. Unlike wind tunnel experiments and field testing, CFD provides details for the velocity and pressure at every point at a given time in the computational domain. Controlled Volume Method (CVM), Finite Difference Method (FDM) and Finite Element Methods (FEM) are a few of the numerical methods used in wind engineering. Of these methods, the FEM method is preferred because it has a higher degree of accuracy. This higher degree of accuracy makes FEM the most reliable method for analyzing the turbulent effects on buildings. Hence, FEM is the method of choice used in this report.

The FCH2D program used in this report is a two-dimensional model based on FEM. The model can be used to analyze the u and v velocity, and the peak pressure at various points within the computational domain. For the purpose of this report, this model was used to investigate the influence of turbulence on building pressure. A single sine wave is considered to simulate turbulence within the model. This is sufficient because it offers a simplified interpretation of the interaction of the wave within the numerical method and building. It also required less computational effort to run the program using only one wave.

The influence of turbulence on building pressure is an issue that requires further investigation. A more systematic study is needed to see the effect of different spectra of turbulence on building pressure. The application of the findings presented in this report is not feasible for use in real life since it is a 2-D model and the turbulence is 3-D in nature. However, the results obtained can be used to refine more complex 3-D models in the field of wind engineering.

1.2 Thesis Objective

The objective of the thesis is to investigate the effects of turbulence on building pressure using the Finite Element Model program developed by Dr. Selvam at the University of Arkansas. Using the FCH2D program the effects changes in flow parameters have on building pressure is studied. Tecplot is used as a visualization tool to evaluate wind flow and the effects that changes in the wavelength and amplitude have on the building pressure at different locations. This objective was achieved through the following steps:

1. Explaining the basis of Finite Element Analysis and understanding how the FCH2D program works
2. Prepare a user manual that describes the input file
3. Run the FCH2D program without a building changing input parameters; wavelength, number of points per wavelength and Reynolds number.
4. Observing the effects changes in input parameters have on the velocity and observing changes in the amplitude
5. Determine a recommended Reynolds number and wavelength that would result in the lowest loss in amplitude.
6. Run the FCH2D program with a building using the recommended Reynolds number and wavelength
7. Investigate the effects of changes in wavelength and amplitude on building pressure
8. Determine the limitations of using the FEM program
9. Provide different applications for which this study would be useful
10. Provide areas in which this study can be developed and the scope for future work

1.3 Organization of Report

This report is organized in the form of chapters. The introduction gives an overview of the project and states the objective of the project. Chapter two presents a literary review of wind engineering, focusing primarily on wind-building interactions. In Chapter 3, a review of finite element computational method is presented with a complete derivation of the different equations used and how they are applicable in the FCH2D model. A preliminary investigation for the transport of turbulence in the model would be found in Chapter 4. The results from the study of the effects of turbulence on building pressure are presented in Chapter 5. Finally, Chapter 6 focuses on the applications of this study and the scope for future work.

CHAPTER 2: LITERATURE REVIEW-BUILDING AERODYNAMICS

2.1 Introduction

Wind Disasters like storms, hurricanes, and tornadoes are responsible for tremendous damage to infrastructure, injury, loss of life and economic damage. Since the early 1980's the intensity, duration and frequency of extreme wind events have increased and are expected to continue increasing in years to come. Therefore, a greater understanding of the wind loadings on buildings is needed.

During the year of 2017, three category 4 hurricanes made landfall in the US causing Billions of dollars in damages and significant loss of lives, as reported by the National Oceanic and Atmospheric Administration (NOAA). In August 2017, Hurricane Harvey made landfall near Rockport, Texas causing widespread damage. Though Harvey's devastation was most pronounced due to the large region of extreme rainfall producing historic flooding across Houston and surrounding areas, there were major damages to infrastructure due to wind forces. Tragedy struck again in September 2017, when Hurricane Maria ripped through Puerto Rico. The high winds caused widespread devastation to Puerto Rico's infrastructure and agricultural facilities. Again in September of hurricane Irma made landfall in Florida. The Florida Keys were heavily impacted, as 25% of buildings were destroyed and 65% were significantly damaged. Irma maintained a maximum sustained wind of 185 mph for 37 hours.

Despite several years of research, there is still not a clear understanding of forces acting on building due to wind turbulence. Current knowledge and standards for designing buildings to withstand wind forces are based mainly in wind tunnel experiments and field testing. Most of the wind-load coefficients in ASCE 7-10 wind standards and building codes of practice had been obtained from research carried out by physical modeling. However, these standards are inefficient in accurately representing extreme wind forces on buildings. Hence, infrastructure may not be adequately designed to withstand high wind forces.

2.2 Wind Tunnel Experiments

Wind tunnel testing has been the long-standing method used to investigate building aerodynamics. These tests involve creating a model of the structure. The models are then tested in a wind tunnel. Sensors placed at various locations on the model and instruments inside the wind tunnel provides data regarding the model's interaction with the wind and measure the instantaneous forces, pressure, and velocity.

The wind-tunnel modeling of the Silsoe Cube conducted at the University of Auckland and documented in Richards et al. (2007) gives an insight into the comparisons between wind tunnel testing and field testing. The modeling of the Silsoe cube was conducted at a scale of 1:40. Results from the Auckland wind tunnel test shows that the wind-tunnel cannot reproduce the complete wind spectrum of the field turbulence. The test was unable to capture the low-frequency part of the spectrum and hence the peak pressure is underestimated. The inability to capture the low-frequency end of the spectra is due to scale limitations within the model. Richards explained that "this deficiency is caused by physical limitations created by the tunnel walls that restrict the maximum eddy size that can exist within the tunnel."

Results from the Silsoe Cube model also unveiled that though the mean pressure for the full-scale and wind tunnel was in agreement, the full-scale peak pressures are markedly larger than those from the wind-tunnel. As shown in Figure 2.1 the full-scale peak pressure at some points was more than 200% greater than the wind-tunnel peak pressure. In ASCE 7-10, the maximum reported Component and Cladding pressure coefficient is -2.8 for buildings with less than a 7% roof slope. However, in the results obtained from the full-scale model of the Silsoe Cube pressure coefficients are recorded at values greater than -5.

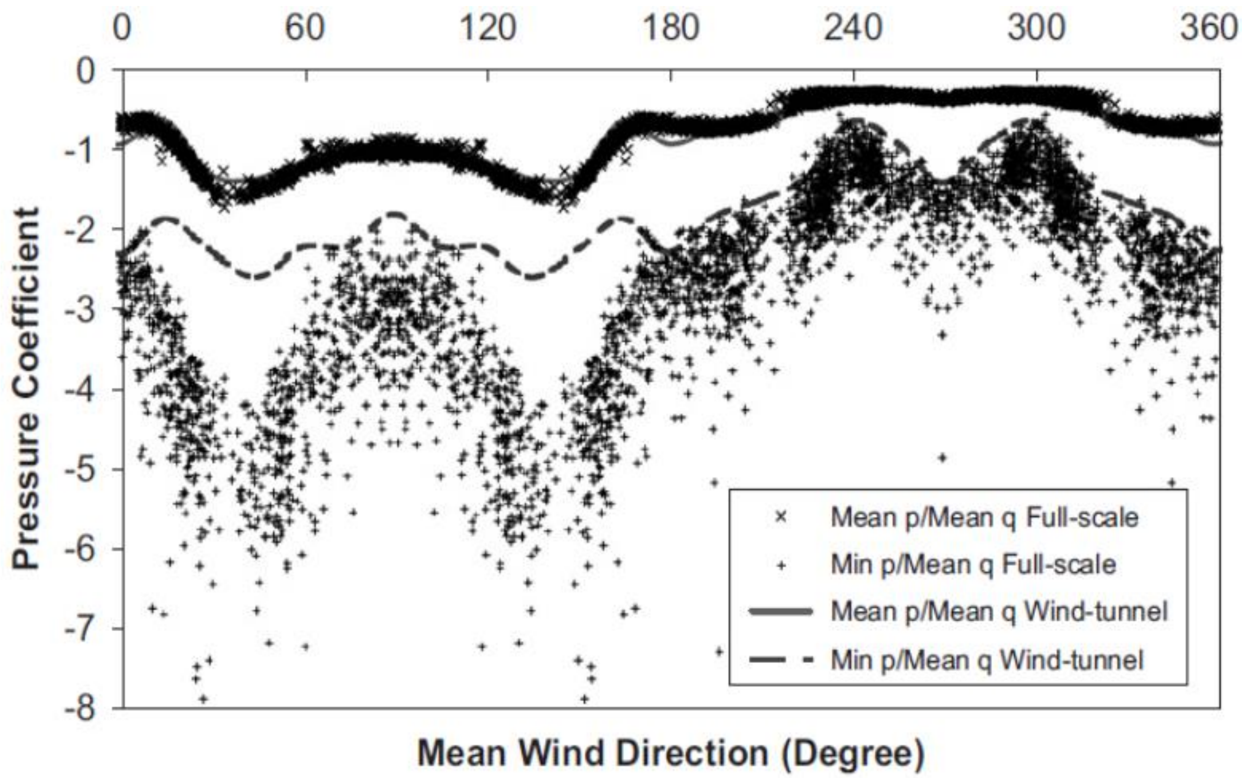


Figure 2.1. *Mean and Peak minimum pressure (Silsoe Cube experiment), Richards et al. (2007)*

Due to these limitations of wind-tunnel testing, it is difficult to model extreme wind events in wind tunnels. It is also expensive and time-consuming. These drawbacks have influenced researchers to look for more effective ways of investigating building aerodynamics. With the advancement in technology, researchers are now investigating the use of Computational Fluid Dynamics as a tool for assessing wind loadings.

2.3 Computational Methods

Computational fluid dynamic is becoming an alternative tool in wind engineering. Extensive developments continue to occur in computational wind engineering due to the availability of high-performance computers and storage systems. CFD can provide instantaneous details of velocity, pressure, and temperature. It is also cost effective and efficient, providing data within hours or days, as opposed to weeks in the case of physical testing. Investing efforts in improving CFD methods could reduce the reliance on expensive and time-consuming physical testing methods.

The capability of CFD models like the Control Volume Method (CVM) and the Finite Difference Method (FDM) for use in building aerodynamics have been investigated. It has been found that these CFD methods cannot reproduce the high-frequency portion of the wind spectrum as shown in Figure 2.2. Hence, like with wind-tunnel testing, the peak pressures are underpredicted.

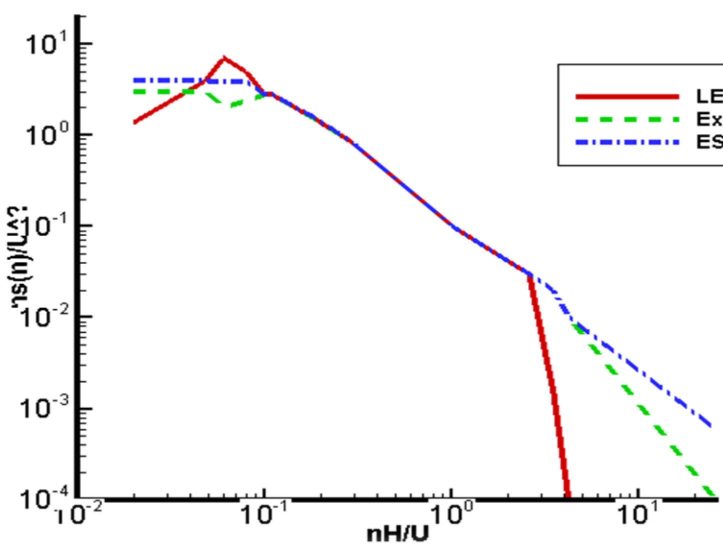


Figure 2.2. *Wind Spectrum (CFD and Physical Testing), Selvam et al (2018)*

A comparative study of CFD methods for different wind engineering problems was documented in Selvam et al. (2018). His works highlighted the challenges in computing turbulent effect on peak pressure on building by investigating the transport of turbulence using a one-dimensional pure convection problem. From his findings, Selvam concluded that lower accuracy method the FDM and CVM require a large number of points per wavelength and a greater number of grid points. He concluded that for preliminary analysis, a minimum of 26 points per wavelength and 4.4×10^9 should be used for efficient transport of the wave when using CVM or FDM. Even with such a large number of grid points, there was still a recorded 70% reduction in the wave amplitude just after traveling 5 wavelengths.

The Finite Element Method is better suited for wind engineering problems because it has a higher degree of accuracy. Therefore, fewer grid points are required which significantly reduces computational time and storage space. As documented in Selvam et al. (2018) the Finite Element method transported the single sine wave at a greater accuracy than the FDM and CVM. The reported loss in amplitude was 20%. It is suggested that 13 points per wave and 3.1×10^7 grid points are needed to accurately transport turbulence. Therefore, this method is preferred for wind engineering problem that involves turbulence and is hence the method used in this report.

CHAPTER 3: COMPUTATIONAL MODEL-
FINITE ELEMENT METHOD

3.1. Introduction

Most of the modeling work in the past have been conducted without inflow turbulence. Selvam et al (1997) reviewed the status of inflow turbulent generation. In his works, he compared the pressure coefficients with measurements obtained from the Texas Tech University physical model.

The two-dimensional FCH2D finite element program used in this work was created by Dr. Selvam of the University of Arkansas. The program is intended to simulate the flow of straight-line boundary layer wind around a rigid square. Three output files are created for each run of the program; velocity vs. time plot, pressure coefficient vs. time plot and a contour plot for pressure, and velocity. In this report, a single sine wave was used at the inlet to represent inflow turbulence within the model. Turbulence is a combination of multiple waves with a wide range of different amplitudes, frequencies, and wavelengths. However, a single sine wave is sufficient in this work because it serves as a benchmark problem in analyzing the effects of turbulence on building pressure.

3.2. Flow parameters

The following parameters that characterize flow are the frequency (f), angular frequency (ω), Reynolds number (R_e), and pressure coefficient (C_p).

$$f = \frac{v}{\lambda} \qquad \text{equation 3.1}$$

$$\omega = \frac{2\pi}{T} \qquad \text{equation 3.2}$$

$$Re = \frac{v.H}{\upsilon} \qquad \text{equation 3.3}$$

$$C_p = \frac{p}{\frac{1}{2} \cdot \rho \cdot v^2} \qquad \text{equation 3.4}$$

Where H is the reference height and also in this case building height, v is the reference velocity at building height, λ is the wavelength, T is the period, p is the pressure and ρ is the density.

3.3. Governing equations

The Navier-Stokes (NS) equations are the governing equations for fluid mechanics problems. The three components of the NS equations are momentum, continuity and internal energy. Finite element methods use the non-conservative form of the NS equation. Details of the derivation of the NS equations can be found in Anderson (1995). The NS equation in tensorial notation is written below. Also, the variable in the subscript with a comma means differentiation.

Continuity equation: $U_{i,i}=0$

Momentum equation: $U_{i,t} + U_j \cdot U_{i,j} = -(p/\rho)_i + [v(U_{i,j} + U_{j,i})]_j$

Where; U_i=velocity in the i^{th} direction, P=pressure and P=fluid density. The turbulence whose wave length less than the grid size within the computational domain is modeled using large eddy simulation. For computations a non-dimensional equation is used.

3.4. Boundary conditions

Boundary conditions are specified at different locations in the computational domain. Inlet and wall nodes are Dirichlet boundaries or velocities are specified. At the walls, the velocities are zero and at the inlet and top boundary, u=1+A*sin(wt) and v=-A*sin(wt). Here the amplitude will be 0.2 or less and the frequency varies depending upon the wavelength. Figure 3.1. shows the boundary conditions at different locations in the model. All parameters were non-dimensionalized with respect to the building height (H), which is taken as 1.

Boundary conditions:

- The velocity at the inlet and the top boundary: x-direction, u = u (t) = 1+A*sin(wt) and y-direction, v = v (t) =-A*sin(wt)
- Velocity at the top bottom boundary is no slip conditions, velocity in the x and y direction are both zero.
- Pressure at the top half of the outlet is zero.

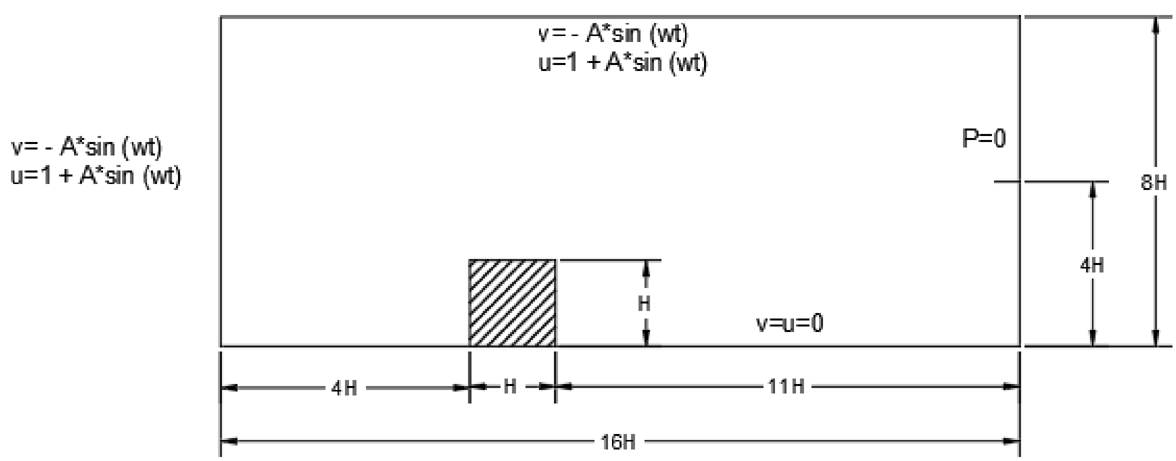


Figure 3.1. *Solution domain and boundary conditions*

3.5. Grid discretization

The computational domain was discretized into equally spaced quadrilateral elements. A grid is defined by η, ξ or I, J system of mesh lines. Where these mesh lines intersect forms a node. In this works, a fixed, structured grid is used. In a structured mesh, all interior nodes are surrounded by equal numbers of adjacent nodes as depicted in Figure 3.2.

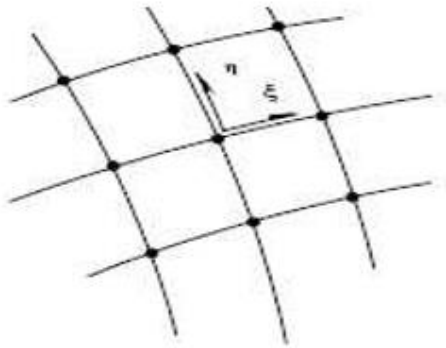


Figure 3.2. *Structured grid discretization, Murray et al. (2007)*

The accuracy with which the model represents the physical body is dependent on the density and distribution of the grid lines. Closer spaced nodes lead to more accurate results. In this works. An equally spaced grid was used so that the transport of turbulence within the model can be accurately captured.

A total of (257 x 129) nodes were used. The building height, which is 1, has a total of 16 points. Therefore, each unit within the model has 16 points. This is important for understanding the relationship between the wavelength of the inflow turbulence and the number of points per wavelength. For example, one wavelength has 16 points per wavelength. To determine the number of points per wavelength, the wavelength should be multiplied by 16, ($\lambda \cdot 16$).

CHAPTER 4: PRELIMINARY INVESTIGATIONS-
TRANSPORT OF SINE WAVE

4.1. Overview

The finite element program was used to investigate the transport of turbulence within the model. As stated in previous chapters, the program computes the velocity and pressure at different locations. A study was conducted without the presence of a building to determine how changes in flow parameters affect the turbulence at the location where a building would later be placed. The percentage loss in amplitude of the turbulence was documented. The building flow model was then analyzed with turbulence and without turbulence to investigate the turbulent effect on building pressure, using these preliminary investigations as a guide.

4.2. The Effects of Wavelength

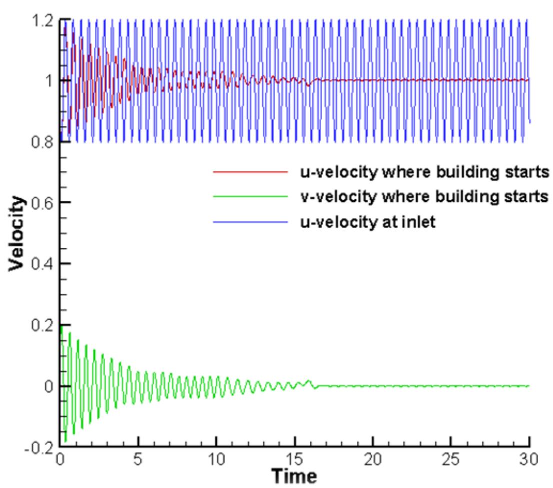
A model was created without a building to investigate the effects the change in wavelength would have on the turbulent velocity. The mean velocity is 1 and the turbulent amplitude is 0.2. All other parameters within the model were kept constant. Each run was conducted with a Reynold Number of 10,000, therefore the viscosity of the flow was 1×10^{-4}. Changes in the wavelength inversely affect the frequency of the wave. The wavelength also governs the number of points per wavelength in the model.

The longer the wavelength the more efficiently the turbulence can be transported within the model. This is due to the fact the longer wavelengths have more grid points per wavelength that can more accurately capture the changes in turbulence. Conversely, the shorter the wavelength the more difficult it is to transport turbulence. The rapid changes in velocity make it difficult to capture changes in addition to there being fewer points per wavelength.

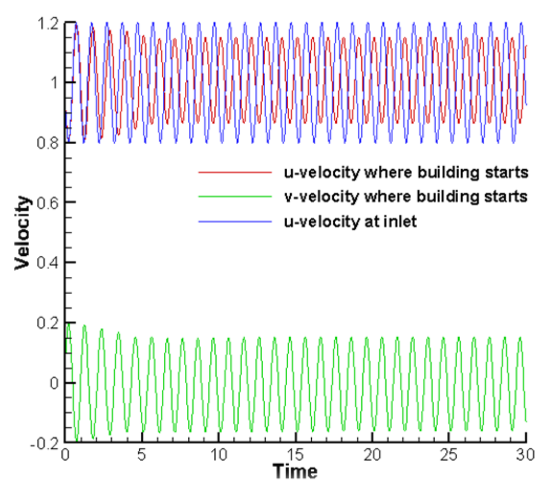
The greatest loss in amplitude was recorded for the flow with the highest frequency or shortest wavelengths. There was almost a 100% reduction in amplitude. However, for higher wavelengths and therefore lower frequencies there is much less loss in amplitude. It can hence be concluded that the finite element model more accurately transports waves with a lower frequency.

Table 4.1 *The percentage loss in amplitude for different wavelengths*

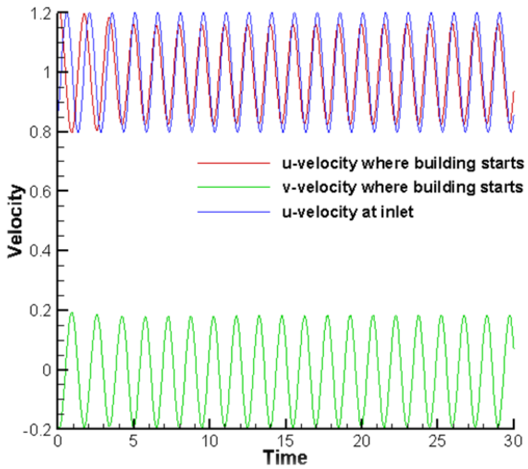
Wavelength	Points per Wavelength	Amplitude	% Reduction
0.5	8	0.005	97.5
1	16	0.150	25
1.5	24	0.161	19.5
2	32	0.198	1



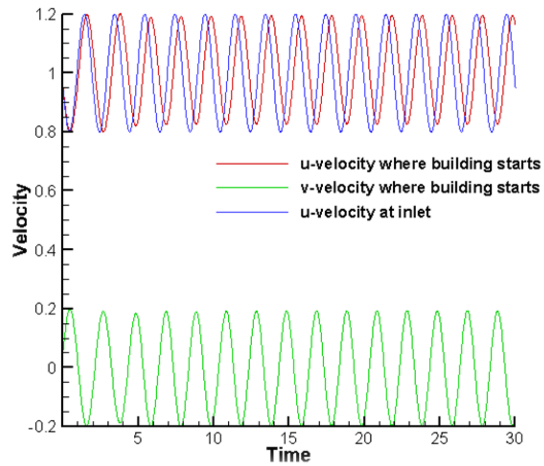
(a).



(b).



(c).



(d).

Figure. 4.1. *The effects of Wavelength on velocity (a) λ=0.5; (b) λ=1; (c)λ=1.5; (d)λ=2 with amplitude of 0.2. (RE= 10,000)*

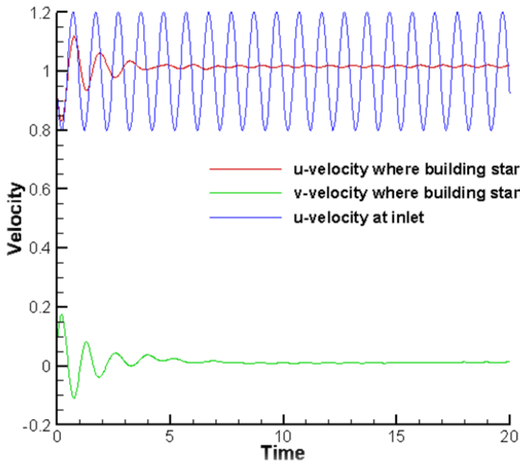
4.3. The Effects of Reynolds Number

The effects a change in Reynolds number was investigated in a model without the building. The mean inlet velocity was 1, the turbulent amplitude was 0.2 and the wavelength was one. All other parameters within the model were kept constant. Changes in the Reynolds Number also governs changes in the viscosity. An increase in Reynolds number causes a decrease in viscosity. The transport of turbulence is also affected by the flow pattern, how turbulent the flow is. The reduction in amplitude is due to Re effect and numerical error. When enough points to transport the waves are there then numerical errors can be reduced.

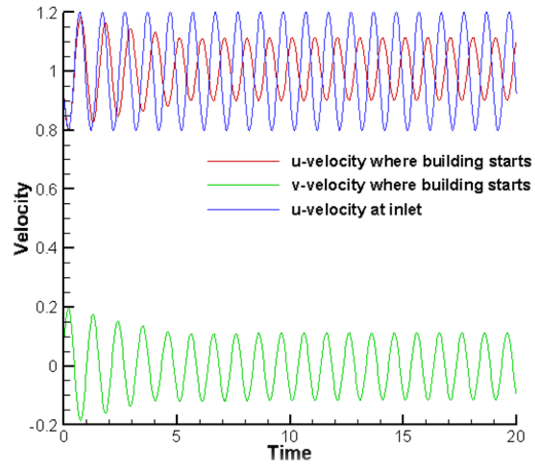
The flow with the lowest Reynolds Number had almost a 90% reduction in amplitude. As the Reynolds Number increased the percent reduction decreased. Hence it can be concluded that as the Reynolds number increases the percent reduction of the turbulent velocity decreases.

Table 4.2. *Percentage Reduction in Turbulent Amplitude due to changes in Reynolds Number*

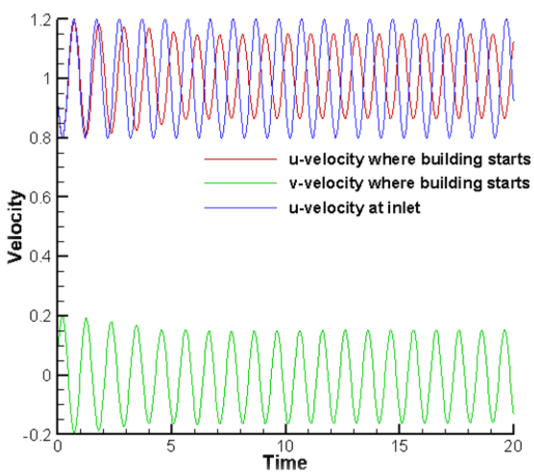
Re	Viscosity	Amplitude	% Reduction
100	0.01	1.021	89.8
1,000	0.001	1.114	43
10,000	0.0001	1.150	25
100,000	0.00001	1.155	22.5



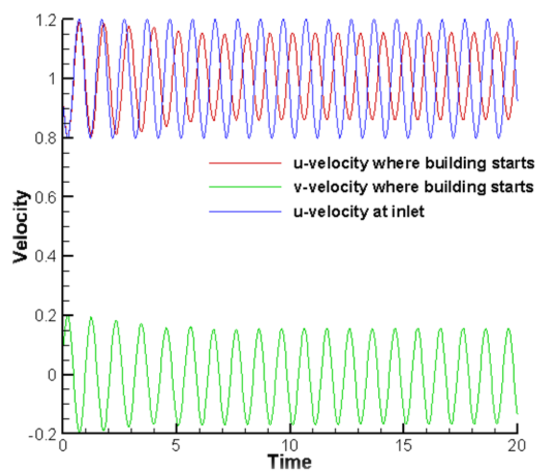
(a).



(b).



(c).



(d).

Figure. 4.2. *The effects of Reynolds Number on velocity (a) Re=100; (b) Re=1,000; (c)Re=10,000; (d)Re=100,000 with amplitude 0.2*

CHAPTER 5: RESULTS

5.1. Overview

Understanding the effects of turbulence on building peak pressure is still a challenge in the Wind Engineering field. Turbulence causes building pressure to fluctuate in very short periods of time. In this work, flow around a square is analyzed to observe the effects of turbulence on building pressure. Changes were made to the inflow wave and the effects these changes have on building pressure, and the formation and transport of vortices were observed. The pressure was observed at the locations shown in Figure 5.1.

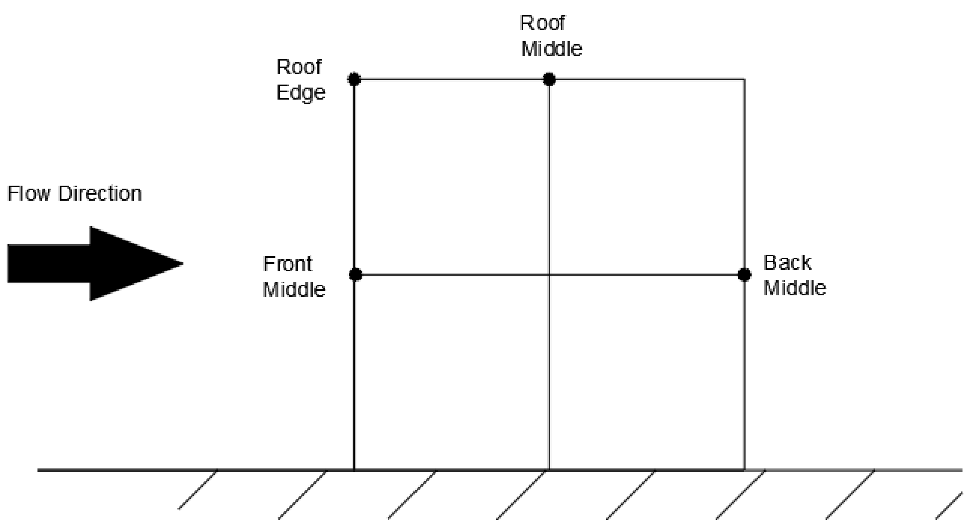
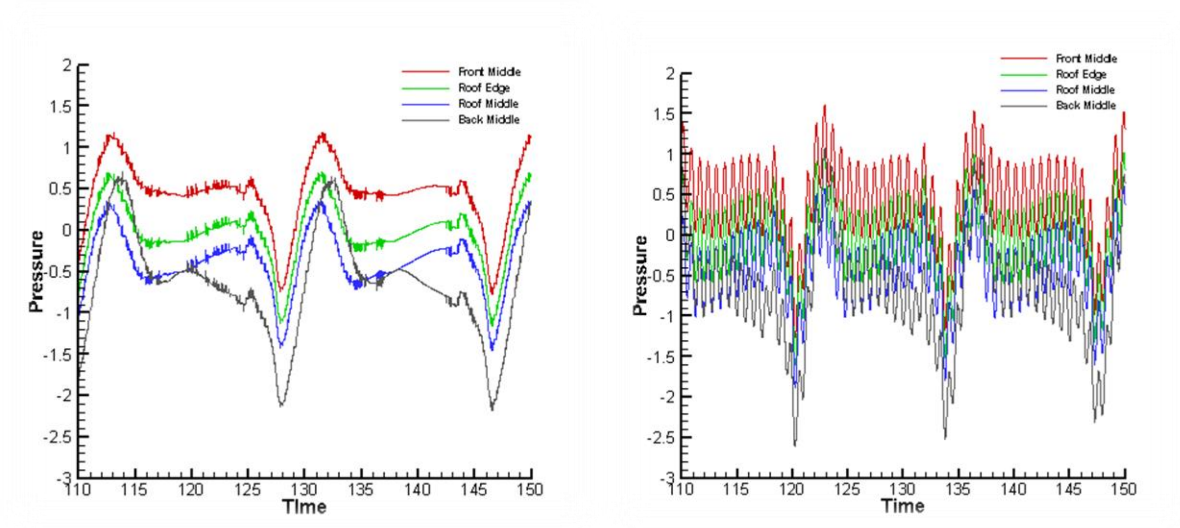


Figure 5.1. *Locations of pressure observations*

5.2. Wavelength effects

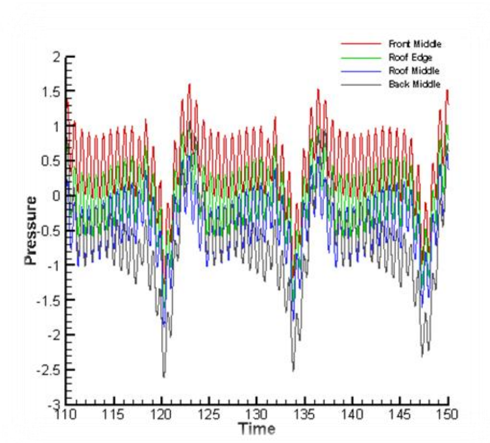
An analysis for variation in wavelength was conducted for flow around the building without turbulence. With no turbulence, the sinusoidal properties of the wave were still developed due to building effect. As shown in Figure 5.2, there was recorded pressure variation in the flow. However, pressure variation occurred much slower than the observed pressure variation in the model with turbulence.

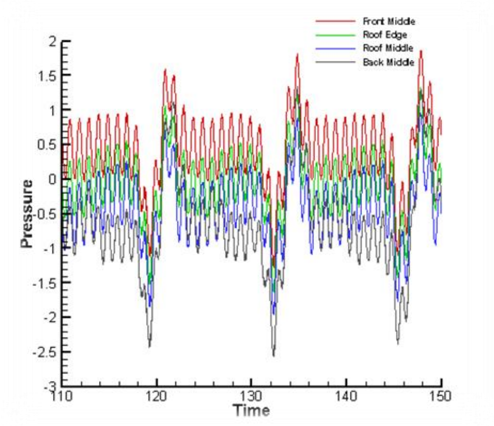


(a). *(b).*

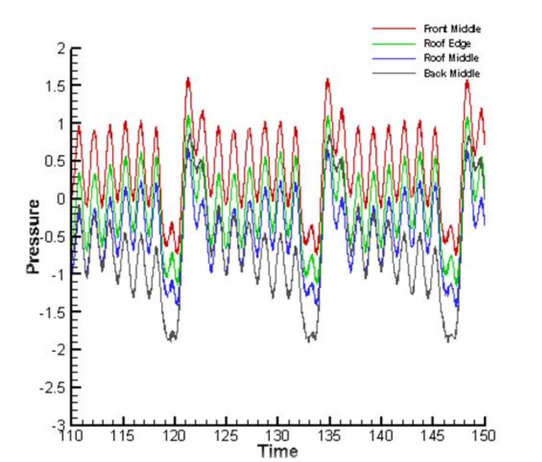
Figure 5.2. *Pressure variation (a) without turbulence and (b) with turbulence (wavelength = 2)*

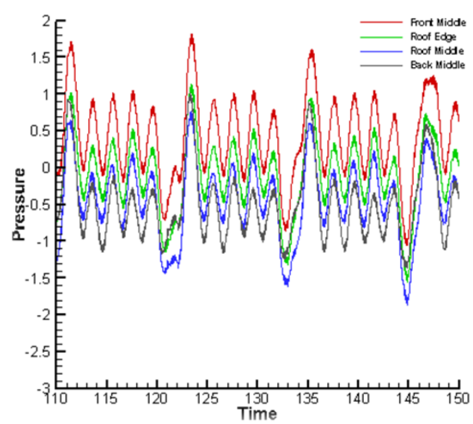
Further analysis was conducted by changing the wavelength on the inflow wave. An amplitude of 2 and a Reynolds number of 10,000 was used. There was greater variation in pressure for the smaller wavelengths than for larger wavelengths. The smaller wavelength also had a higher recorded peak pressure. The pressure variation observed with changes in wavelength is affected by the frequency of the wave. Higher frequency waves have higher pressure variation than lower frequency waves. From the plots in Figure 5.2 it is evident that the negative peak pressure are greater when the wavelength is less than the building height.





(a). *(b).*



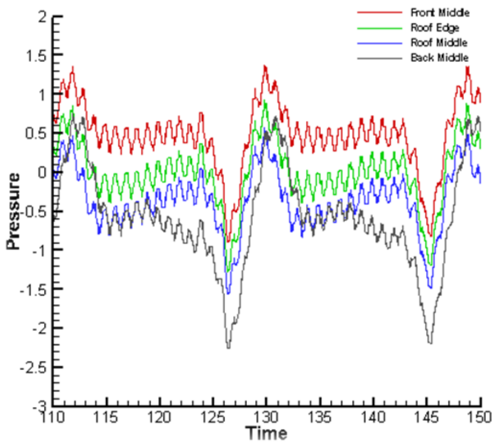


(c). *(d).*

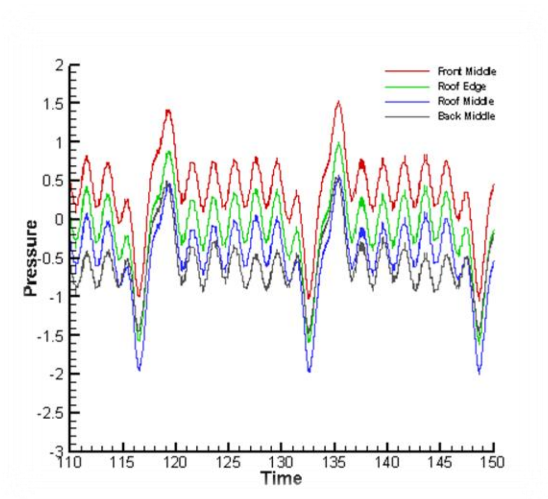
Figure 5.3 *Pressure variation with changes in wavelength (a) wavelength 0.75 (b) wavelength 1, (c) wavelength 1.5 (d) wavelength 2*

5.3. Amplitude effects

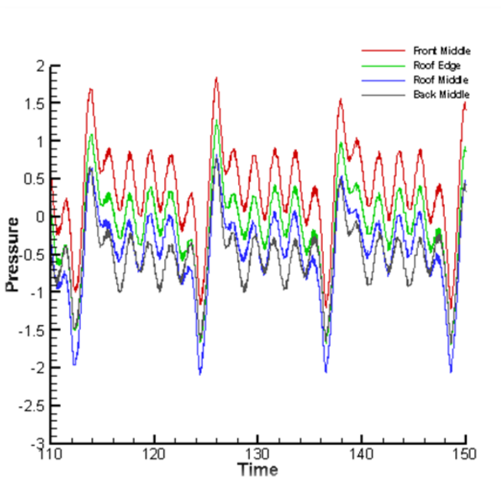
The effects of changes in amplitude was also investigated. A wavelength of two and Reynolds number of 10,000 was used in this study. From the results obtained the waves with the smaller amplitude have a higher negative pressure. There was also more pressure variation for the waves with smaller amplitudes. It was anticipated that the waves with larger amplitudes would have larger pressure values. However, this I not the case as shown in Figure 5.4.



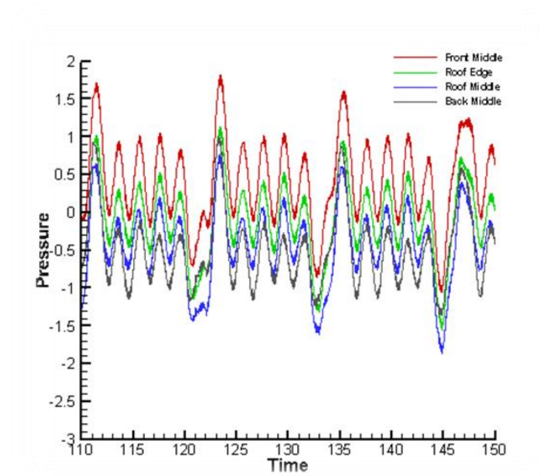
(a).



(b).



(c).

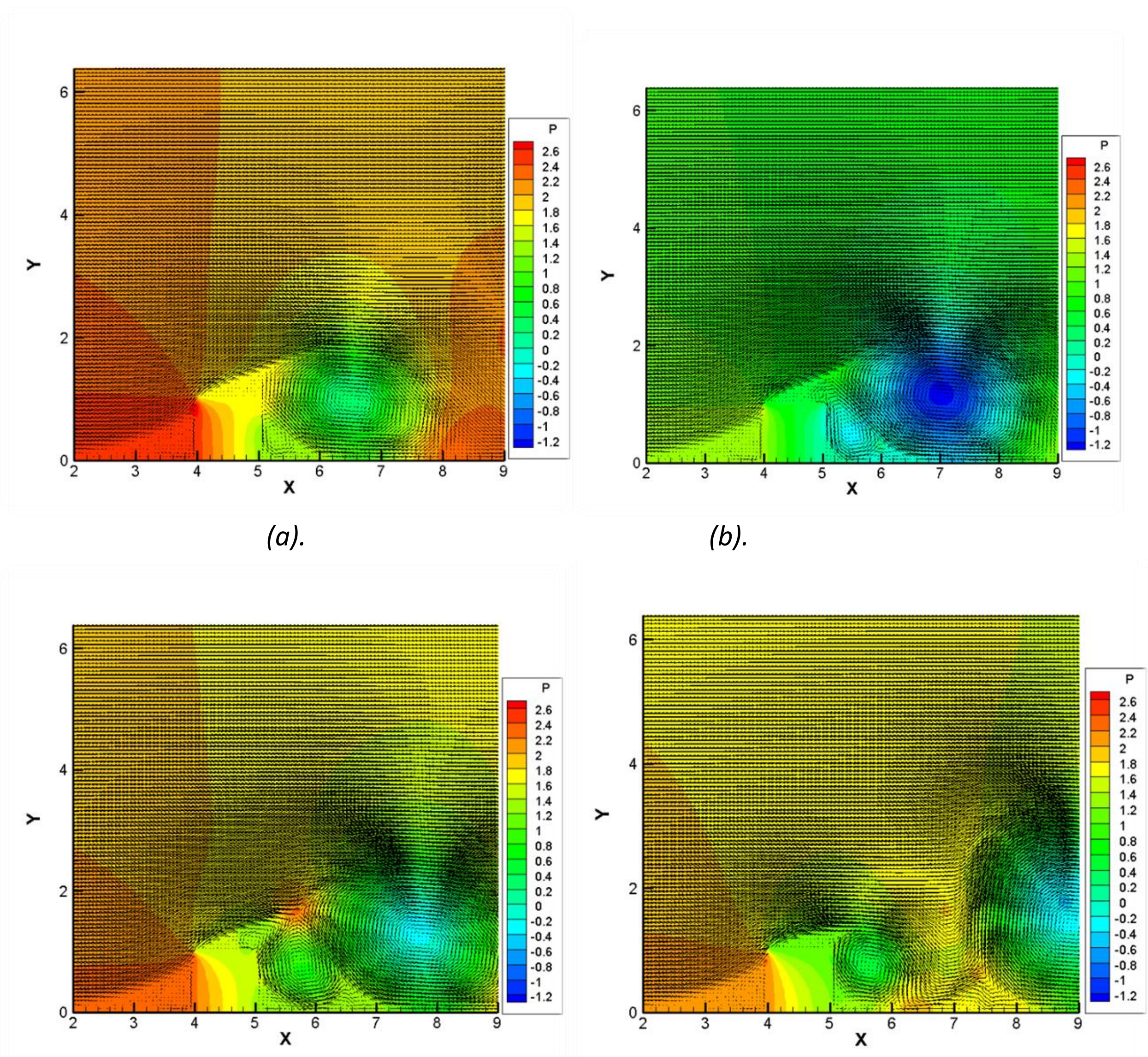


(d).

Figure 5.4 *Pressure variation with changes in amplitude (a) amplitude 0.05 (b) amplitude 0.1, (c) amplitude 0.15 (d) amplitude 0.2, (wavelength = 2)*

5.3. Contour Plots:

The formation, intensity and transport of vortices in the model also influences the building pressure. The vortices formed in the model without turbulence are less intense and are quickly swept away from the building. This causes less pressure variation and lower peak pressure

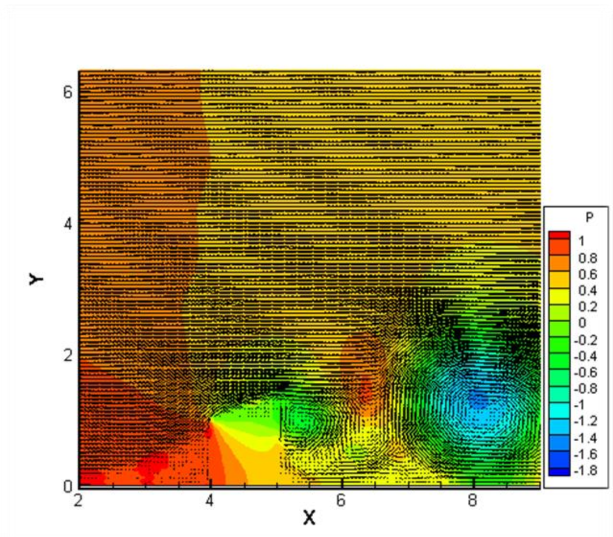


(a). (b).

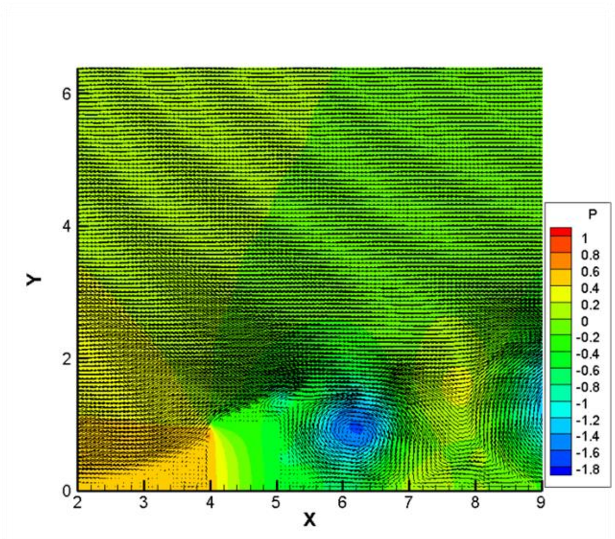
(c). (d).

Figure 5.5. *Vortex progression without turbulence*

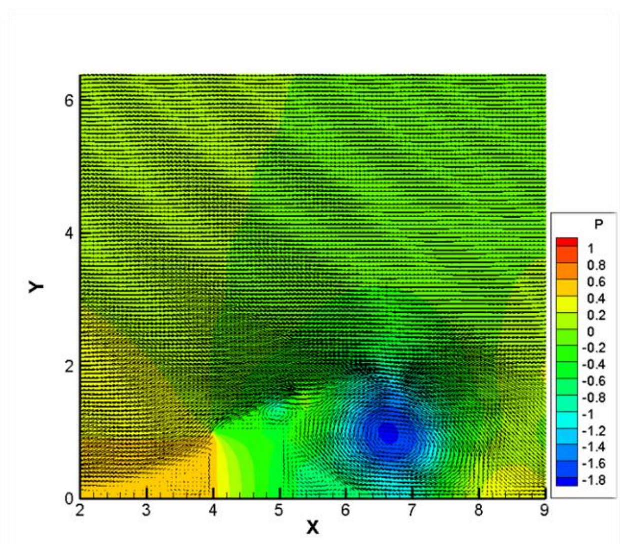
When turbulence in introduced in the model, the vortices formed are more intense as shown in Figure 5.6. The vortices are also transported much slower with turbulence which results in higher recorded peak pressure.



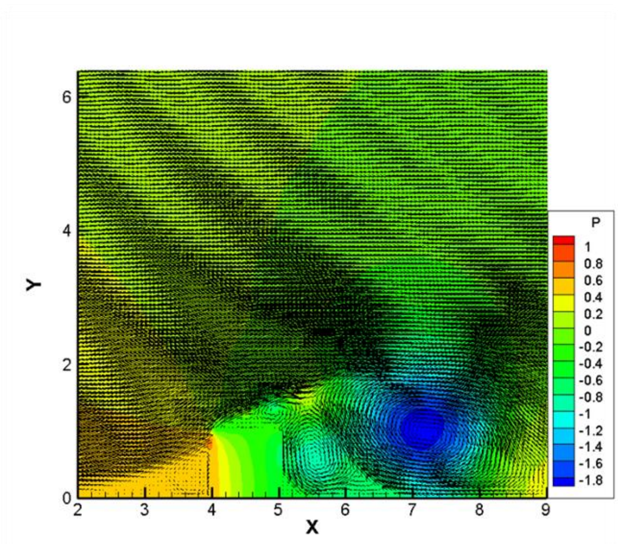
(a).



(b).



(c).



(d).

Figure 5.6. *Vortex progression with turbulence, wavelength 2, amplitude 0.1*

When turbulence is introduced into the model, the transport of the vortices is influenced by the wavelength and amplitude of the wave. For smaller wavelengths the vortices are transported at a much slower rate. The cyclic motion of the vortices induces changes in pressure in addition to the sinusoidal motion of the wave. Higher pressures are recorded when the vortices are closer to the building. The rate at which the vortices move away from the building also influences the pressure variation. The longer the vortex take to move away from the building the more pressure variation is recorded. Waves with smaller amplitudes and smaller wavelengths result in slower transport of vortices and hence greater pressures are recorded.

CHAPTER 6: CONCLUSION

6.1. Summary

Most of the modeling work thus far has been conducted without turbulence. Turbulence causes the velocity to change rapidly which may in turn affect the building pressure. In this work, a single sine wave was used to represent turbulence in the two-dimensional finite element program. The effects of these changes on building pressure was observed.

The results obtained from this study shows that there are four major components that affect building pressure in the finite element model. These factors include; the mean velocity, turbulent intensity (amplitude), wavelength and the transport of the vortices formed. Ultimately, the formation and transport of vortices on and around the building accounts for the pressure variations obtained. However, flow properties like the wavelength and the amplitude dictates how these vortices are transported. Shorter wavelengths result in a slower transport of the vortices which results in more pressure variation on the building and slightly higher peak pressures. The same is true for waves with shorter amplitudes.

6.2. Recommendations for Future Work

- Future works should be done using multiple waves to better represent turbulence.

REFERENCES

Anderson, J.D. (1995). "Computational Fluid Dynamics, The Basics with applications", McGraw Hill, 43-78.

ASCE (2010) Minimum Design Loads for Buildings and Other Structures. ASCE/SEI Standard 7-10

Murray, S. (2007). Validation of adaptive aeroelastic finite element program for two dimensional flow past a suspended bridge girder section using unstructured grid. (PDF File). Retrieved from http://scholarworks.uark.edu/cveguht

Nozawa, K., Tamura. (2002). Large eddy simulation of flow around a low-rise building immersed in a rough wall turbulent boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics*, 90, 1151-1162

Richards, P.J., Hoxey, R.P., Connell, B., Lander, D. (2007). Wind-tunnel modelling of the Silsoe Cube. *Journal of Wind Engineering and Industrial Aerodynamics*, 95, 1384-1399.

Selvam, R. P. (1997). Computation of pressure on Texas Tech University building using large eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics*, 67, 68, 647-657.

Selvam, R. P. (2017). CFD as a tool for assessing wind loading. *The Bridge and Structural Engineer*. 47, 1-6.

Selvam, R. P., (2017) *Introduction to Computational Fluid Dynamics and Heat Transfer*.

Selvam, R. P., Chowdhury, A., Irwin, P. (2018). CFD Performance for Different Wind Engineering Problems, The 1st International Symposium on Computational Wind Engineering.

Selvam. R. P., (2010), *Building and Bridge Aerodynamics using Computational Wind Engineering*

Smith, A., Lott, N., Houton, T., et al. (2015) U.S. Billion-Dollar Weather and Climate Disasters 1980-2019 (PDF file). Retrieved from https://www.ncdc.noaa.gov/billions/events.pdf

Stathopoulos, T. (1997). Computational wind engineering: Past achievements and future challenges, *Journal of Wind Engineering and Industrial Aerodynamics*, 67 & 68, 509-532

Yang, W., Quan Y., Jin X., Tamura Y., Gu M. (2008). Influences of equilibrium atmosphere boundary layer and turbulence parameter on wind loads of low rise buildings, *Journal of Wind Engineering and Industrial Aerodynamics*, 96, 2080-2092

APPENDIX – A: USER MANUAL FOR FINITE ELEMENT PROGRAM-FCH.2D

USER MANUAL FOR FINITE ELEMENT PROGRAM-FCH.2D

The FCH.2D program is used to study effects of turbulence on building pressures. This user manual provides details on the preparation of the input data and offers an interpretation of the output data.

1. **Input Line 1:**

READ(3,*)IM,JM,XL,YL,IB1,IB2,JB1,JB2
IM: Number of points in the X direction
JM: Number of points in the Y direction
XL: Domain Length in X
YL: Domain Length in Y
IB1: Beginning of I for building
IB2: End of I for building
JB1: Beginning of J for building
JB2: End of J for building

2. **Input Line 2:**

READ(3,*)VISC,DT,TTIME,utum,WLEN,imovie,IREAD
VISC: Viscosity (1/Reynolds Number)
DT: Time Step
TTIME: Total time to run
UTUM: Turbulent amplitude
WLEN: Wavelength considered
IMOVIE: Time steps after movie files are written
IREAD: =0 write file, =1 read the data but do not write file

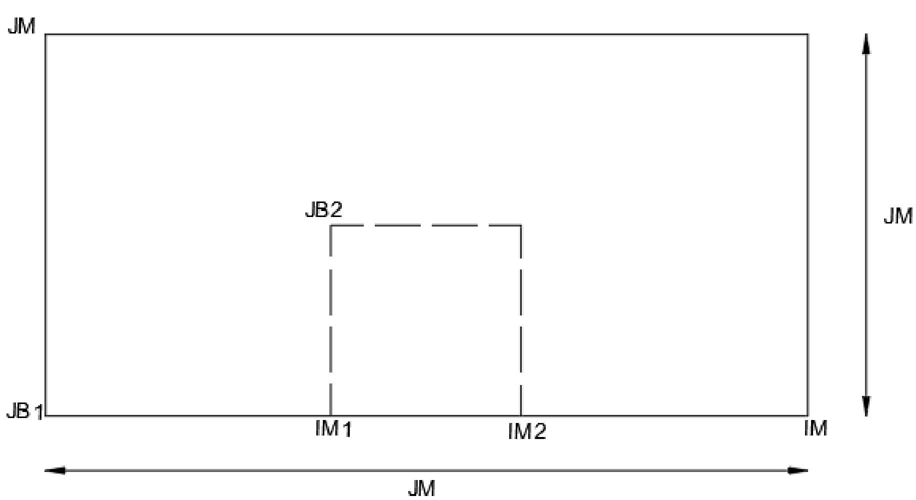


Fig.1. *Schematic Layout of Input Data*

In the input file fcha-i.txt. the variables IM and JM governs the number of grid point used. The total number of grid points is given by their product (IM x JM). The total number of grid point used is (257 x 129)

3.Output Data

Fcha-o.plt: This is a velocity vs. time plot. It writes the u-velocity where the building start, v-velocity where the building starts and the u-velocity at the inlet. All velocities as analyzed at the middle of the grid (jm/2)

Fcha-o2.plt: This is a pressure vs time plot.

Fcha-p.plt: This is the final output for plotting

APPENDIX – B: INSTRUCTIONS FOR MAKING ANIMATION DATA IN TIME

Making animation of a data in time

```
#!MC 800
$!VarSet |NumFiles| = (23).1
$!EXPORTSETUP EXPORTFORMAT = AVI
$!ExportSetup ExportFName =('D:\rps\f18\research\cwe\building-fem\fem2d-building\flow-
re1000\b2d.avi')2
$!Loop |NumFiles|
$!OpenLayout ('D:\rps\f18\research\cwe\building-fem\fem2d-building\flow-re1000\b2d.lay')3
  AltPlotFNames = ('D:\rps\f18\research\cwe\building-fem\fem2d-building\flow-
re1000\mv|LOOP|.plt')4
$! IF |LOOP|==1
  $!EXPORT
    APPEND =NO
$!ENDIF
$! IF |LOOP| !=1
  $!EXPORT
   APPEND =yes
$!ENDIF
$!Endloop
$!quit
```

1. Give the # of movie files you have instead of 23
2. Give path to store b2d.avi movie file or any other name you give
3. Give path to read the layout file (here it is b2d.lay).
4. Give path to read the mv files