WIND TUNNEL BLOCKAGE CORRECTIONS:

A COMPUTATIONAL STUDY

by

DEEPAK SAHINI, B.Tech.

A THESIS

IN

MECHANICAL ENGINEERING

Submitted to the Graduate Faculty
of Texas Tech University in
Partial Fulfillment of
the Requirements for
the Degree of

MASTER OF SCIENCE

IN

MECHANICAL ENGINEERING

Approved

______________________________
Chairperson of the Committee

______________________________

Accepted

______________________________
Dean of the Graduate School

August, 2004
ACKNOWLEDGMENTS

I consider this as a privilege to thank the people who have made this thesis possible.

I would like to sincerely thank my advisor, Dr. Siva Parameswaran for his support, encouragement, and assistance during my work. He, constantly monitored my progress and provided me with a great deal of ideas throughout my thesis work.

I thank Dr. Darryl L. James and Dr. Padmanabhan Seshaiyer for serving on my committee. I appreciate the help and assistance from my colleagues, Dr. Senthooran Sivapalan and Dr. Dong Dae Lee. I thank the Department of Mechanical Engineering for the excellent CFD lab facility. I also appreciate the services rendered by the Texas Tech Library for providing assistance with the necessary literature.

Now, I take the opportunity and pride of thanking my father Balagangadhar Tilak Sahini, mother Jayalakshmi Sahini and my sister Shilpa Rekha Sahini, without whose support and motivation this achievement of mine, would not have been possible. Last but not least, I thank my roommates and friends, Venkata Suman Movva, Seethapathi Rao Latchireddi, Sandeep Singh Thakur, Anil Kumar Chandolu, Ravi Sekhar Gunda, Niranjan Das Mettu, Pavan Chintapanti, Ram Prashant Nellore and Kishore Kumar Gali for their valuable suggestions, encouragement and cooperation during all times.
# TABLE OF CONTENTS

ACKNOWLEDGMENTS ......................................................................................... ii

ABSTRACT ................................................................................................. v

LIST OF TABLES .......................................................................................... vi

LIST OF FIGURES ......................................................................................... vii

NOMENCLATURE .......................................................................................... xii

CHAPTER

1 INTRODUCTION ......................................................................................... 1

2 BACKGROUND WORK AND LITERATURE REVIEW ......................... 5

   2.1 Wind tunnel Testing and Blockage Corrections ......................... 5

   2.1.1 An Introduction to Wind Tunnel and its General Layout ........ 5

   2.1.2 Wind Tunnel Blockage Corrections ....................................... 8

   2.1.3 Basic Mechanism associated with Blockage Effects ............ 9

   2.1.4 Role of CFD in Wind Tunnel Blockage Corrections ............ 11

   2.2 Literature Review ........................................................................... 12

   2.2.1 Review of work on Experimental determination of Blockage Correction .................................................. 12

   2.2.2 Review of work on Computational determination of Blockage Correction ............................................. 16

3 COMPUTATIONAL METHODOLOGY ....................................................... 21

   3.1 Assumptions .................................................................................. 22

   3.2 Computational Setup and Solution Procedure ......................... 22
3.2.1 Domain Layout Considerations ........................................... 22
3.2.2 Mesh Generation ................................................................. 27
3.2.3 Boundary and Initial Conditions ......................................... 39
3.2.4 Turbulence Models and Numerical Methods ............................ 45

4 RESULTS AND DISCUSSION ......................................................... 51

4.1 Case (A) Two Dimensional Flow around a rectangular section
subjected to an unconstrained flow field ...................................... 52

4.2 Case (B) Two Dimensional Flow around a rectangular section
subjected to a constrained flow field in a wind tunnel test section .... 63

4.3 Case (C) Three dimensional flow around a rectangular block
placed on the floor in a wind tunnel test section ............................ 74

4.4 Case (D) Three Dimensional flow around a top-front edge
tapered rectangular blockage placed close to the floor of a
wind tunnel test section ............................................................ 85

4.5 Grid Refinement Study on Case (C) for 10% Blockage Ratio ........ 100

5 CONCLUSIONS AND FUTURE WORK .......................................... 105

REFERENCES .............................................................................. 107

APPENDIX

A. VECTOR PLOTS FROM PHOENICS ........................................... 110

B. VECTOR PLOTS FROM AIRFLO3D ........................................... 116
Wind tunnel blockage testing has been a widespread traditional practice in the automobile industry for many years; but the tests conducted have been associated with the so-called blockage effects, which arise due to the constrained flow nature inside the wind tunnel test section and over the blockages. These blockage effects need to be corrected in order to comprehend the test results similar to those of the actual road conditions. CFD has emerged as a tool to determine the blockage effects and provide corrections using computational techniques.

In this present study, two such CFD packages, namely PHOENICS and AIRFLO3D, are used for determining the wind tunnel blockage effects. The problems taken into consideration are both two dimensional and three dimensional flow cases. The test section domain height is varied so as to produce different blockage ratios, keeping the blockage dimensions constant. A two dimensional free stream case with blockage \( \frac{l}{h} \) ratio variation is tested and compared with experimental results. In the other cases, the two packages are compared with each other for pressure and velocity distributions and drag coefficients. A grid independent study was performed for one case. Finally, blockage correction equations are obtained for all the test cases.
## LIST OF TABLES

3.1 Magnitude of \( l \) for different \( l/h \) ratios ................................................................. 24

3.2 Height of the wind tunnel and the vertical location of the blockage for 5\%, 10\% and 15\% blockage ratios ................................................................. 24

3.3 Test section height variation with blockage ratio ................................................. 27

3.4 The mesh sizes of the computational domain for case (A), taking into account the blending factor \( \alpha \) ................................................................. 38

3.5 The mesh sizes of the computational domain for cases (B), (C) and (D), taking into account the blending factor \( \alpha \) ................................................................. 38

4.1 Mesh sizes of Domain of Case (C) for 10\% Blockage for Grid Refinement Study ................................................................. 100
LIST OF FIGURES

2.1. A typical Closed Circuit wind tunnel.........................7

2.2. A typical Open Circuit wind tunnel..............................7

2.3. Plot for Velocity and Pressure Distributions in a Wind Tunnel Test Section due to the effect of Solid Blockage......................9

2.4. Plot for Velocity and Pressure Distributions in a Wind Tunnel Test Section due to the effect of Wake Blockage..........................10

2.5. Plot for Velocity and Pressure Distributions in a Wind Tunnel Test Section due to the combined effect of Solid and Wake Blockage Components..............10

3.1. Layout of the free stream domain with blockage; height of the blockage, h=400mm..............................................23

3.2. Layout of the wind tunnel test section with the rectangular blockage, h=400mm..........................................................24

3.3. Orthographic views of the Wind tunnel and the rectangular blockage for three dimensional flow analysis.........................................26

3.4. Orthographic views of the wind tunnel and the top front edge tapered rectangular blockage, for three dimensional flow analysis...........................26

3.5. Diagram of the mesh generated in ICEM CFD for the case (A). (a) Isometric View. (b) Side View........................................30

3.6. Diagram of the Mesh generated in ICEM CFD for the case (B). (a) Isometric View. (b) Side View........................................31

3.7. Diagram of the mesh generated in ICEM CFD for the case (C). (a) Isometric View. (b) Side View........................................32

3.8. Diagram of the mesh generated in ICEM CFD for the case (D). (a) Isometric View. (b) Side View........................................33

3.9. Diagram of the mesh generated in PHOENICS VR Editor, for the case (A). (a) Side View. (b) Front View.................................34
3.10 Diagram of the mesh generated in PHOENICS VR Editor, for the case (B).
(a) Side View. (b) Front View .................................................................35

3.11 Diagram of the mesh generated in PHOENICS VR Editor, for the case (C).
(a) Side View. (b) Front View .................................................................36

3.12 Diagram of the mesh generated in PHOENICS VR Editor, for the case (D).
(a) Side View. (b) Front View .................................................................37

3.13 Boundary conditions used in ICEM CFD for flow in case (A).
(a) Side View. (b) Front View .................................................................39

3.14 Boundary conditions used in ICEM CFD for flow in case (B).
(a) Side View. (b) Front View .................................................................40

3.15 Boundary conditions used in PHOENICS for flow in cases (A) and (B).
(a) Side View. (b) Front View .................................................................41

3.16 Boundary conditions used in ICEM CFD for flow in case (C).
(a) Side View. (b) Front View .................................................................42

3.17 Boundary conditions used in PHOENICS for flow in case (C).
(a) Side View. (b) Front View .................................................................43

3.18 Boundary conditions used in ICEM CFD for flow in case (D).
(a) Side View. (b) Front View .................................................................44

3.19 Boundary conditions used in PHOENICS for flow in case (D).
(a) Side View. (b) Front View .................................................................45

4.1 Line along which the Pressure and Velocity Distributions are obtained in Two Dimensional case .......................................................52

4.2 Plot for Pressure Signature using PHOENICS data for Case (A) ..........54

4.3 Plot for Velocity Profile using PHOENICS data for Case (A) ..............55

4.4 Plot for Drag variation with $l/h$ ratio using Experimental data for Case (A) ....57

4.5 Plot for Drag variation with $l/h$ ratio using PHOENICS data for Case (A) ....58

4.6 Plot for Drag variation with $l/h$ ratio using AIRFLO3D data for Case (A) ....59

4.7 Plot for Drag Correction using PHOENICS data for Case (A) ..............60
4.8 Plot for Drag Correction using AIRFLO3D data for Case (A) .........................61
4.9 Plot for Pressure Signature using PHOENICS data for Case (B) ..................65
4.10 Plot for Velocity Profile using PHOENICS data for Case (B) ...................66
4.11 Plot for Pressure Signature using AIRFLO3D data for Case (B) ............67
4.12 Plot for Velocity Profile using AIRFLO3D data for Case (B) .................68
4.13 Plot for Drag Coefficient variation with blockage ratio using PHOENICS data..71
4.14 Plot for Drag Coefficient variation with blockage ratio using AIRFLO3D data..72
4.15 Plot for Drag Correction using AIRFLO3D and PHOENICS data for Case (B) ..73
4.16 Line along which the Pressure and Velocity Distributions are obtained in case of Three Dimensional rectangular block .........................74
4.17 Plot for Pressure Signature using PHOENICS data for Case (C) ..............77
4.18 Plot for Velocity Profile using PHOENICS data for Case (C) ..................78
4.19 Plot for Pressure Signature using AIRFLO3D data for Case (C) ..............79
4.20 Plot for Velocity Profile using AIRFLO3D data for Case (C) ..................80
4.21 Plot for Drag Coefficient varying with Blockage Ratio using PHOENICS data..83
4.22 Plot for Drag Coefficient varying with Blockage Ratio using AIRFLO3D data..84
4.23 Lines along which the Pressure and Velocity Distributions are obtained in case of Three Dimensional top-edge tapered block .........................85
4.24 Plot for Pressure Signature at the top of the blockage, using PHOENICS data for Case (D) ........................................88
4.25 Plot for Velocity Profile at the top of the blockage, using PHOENICS data for Case (D) ........................................89
4.26 Plot for Pressure Signature at the top of the blockage, using AIRFLO3D data for Case (D) ........................................90
4.27 Plot for Velocity Profile at the top of the blockage, using AIRFL03D data for Case (D) .................................................................91
4.28 Plot for Pressure Signature at the bottom of the blockage, using PHOENICS data for Case (D) .................................................................93
4.29 Plot for Velocity Profile at the bottom of the blockage, using PHOENICS data for Case (D) .................................................................94
4.30 Plot for Pressure Signature at the bottom of the blockage, using AIRFL03D data for Case (D) .................................................................95
4.31 Plot for Velocity Profile at the bottom of the blockage, using AIRFL03D data for Case (D) .................................................................96
4.32 Plot for Drag Coefficient variation with Blockage Ratio using PHOENICS data .................................................................98
4.33 Plot for Drag Coefficient variation with Blockage Ratio using AIRFL03D data .................................................................99
4.34 Plot for variation of Pressure Signature with Mesh Refinement using PHOENICS Data .................................................................101
4.35 Plot for variation of Velocity Profile with Mesh Refinement using PHOENICS Data .................................................................102
4.36 Plot for variation of Pressure Signature with Mesh Refinement using AIRFL03D Data .................................................................103
4.37 Plot for variation of Velocity Profile with Mesh Refinement using AIRFL03D Data .................................................................104
A.1 Two dimensional rectangular block; 5% Blockage ratio ..........110
A.2 Two dimensional rectangular block; 10% Blockage ratio ..........111
A.3 Two dimensional rectangular block; 15% Blockage ratio ..........111
A.4 Three dimensional rectangular block; 5% Blockage ratio ..........112
A.5 Three dimensional rectangular block; 10% Blockage ratio ..........113
A.6 Three dimensional rectangular block; 15% Blockage ratio ..........113
<table>
<thead>
<tr>
<th></th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A.7</td>
<td>Three dimensional top-front edge tapered block; 5% Blockage ratio</td>
<td>114</td>
</tr>
<tr>
<td>A.8</td>
<td>Three dimensional top-front edge tapered block; 5% Blockage ratio</td>
<td>115</td>
</tr>
<tr>
<td>A.9</td>
<td>Three dimensional top-front edge tapered block; 15% Blockage ratio</td>
<td>115</td>
</tr>
<tr>
<td>B.1</td>
<td>Two dimensional rectangular block; 5% Blockage ratio</td>
<td>116</td>
</tr>
<tr>
<td>B.2</td>
<td>Two dimensional rectangular block; 10% Blockage ratio</td>
<td>117</td>
</tr>
<tr>
<td>B.3</td>
<td>Two dimensional rectangular block; 15% Blockage ratio</td>
<td>117</td>
</tr>
<tr>
<td>B.4</td>
<td>Three dimensional rectangular block; 5% Blockage ratio</td>
<td>118</td>
</tr>
<tr>
<td>B.5</td>
<td>Three dimensional rectangular block; 10% Blockage ratio</td>
<td>119</td>
</tr>
<tr>
<td>B.6</td>
<td>Three dimensional rectangular block; 15% Blockage ratio</td>
<td>119</td>
</tr>
<tr>
<td>B.7</td>
<td>Three dimensional rectangular block; 15% Blockage ratio</td>
<td>120</td>
</tr>
<tr>
<td>B.8</td>
<td>Three dimensional top-front edge tapered block; 10% Blockage ratio</td>
<td>121</td>
</tr>
<tr>
<td>B.9</td>
<td>Three dimensional top-front edge tapered block; 15% Blockage ratio</td>
<td>121</td>
</tr>
</tbody>
</table>
### NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Projected area of cross section</td>
</tr>
<tr>
<td>$A_w$</td>
<td>Area of cross section of blockage</td>
</tr>
<tr>
<td>$A_v$</td>
<td>Area of cross section of wind tunnel</td>
</tr>
<tr>
<td>$C_l$, $C_m$, $C_d$</td>
<td>Model constants of K-ε model</td>
</tr>
<tr>
<td>$C_D$, $C_d$</td>
<td>Drag coefficient</td>
</tr>
<tr>
<td>$C_{Dc}$</td>
<td>Corrected drag coefficient</td>
</tr>
<tr>
<td>$\Delta C_d$</td>
<td>Change in drag coefficient</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Pressure coefficient</td>
</tr>
<tr>
<td>$D_{WB}$</td>
<td>Distance between the NEP and the base of vehicle windshield</td>
</tr>
<tr>
<td>$h$</td>
<td>Height of the blockage placed in the wind tunnel</td>
</tr>
<tr>
<td>$H$</td>
<td>Height of the wind tunnel / the height of the free stream domain considered</td>
</tr>
<tr>
<td>$k$</td>
<td>Distance from the nozzle exit point to the blockage</td>
</tr>
<tr>
<td>$K$</td>
<td>Kinetic Energy of the fluid</td>
</tr>
<tr>
<td>$l$</td>
<td>Length of the blockage placed in the wind tunnel</td>
</tr>
<tr>
<td>$L$</td>
<td>Length of the wind tunnel</td>
</tr>
<tr>
<td>NEP</td>
<td>Nozzle exit point</td>
</tr>
<tr>
<td>$P$</td>
<td>Mean pressure</td>
</tr>
<tr>
<td>$P_i$</td>
<td>Instantaneous pressure</td>
</tr>
<tr>
<td>$P_k$</td>
<td>Production of turbulence kinetic energy</td>
</tr>
<tr>
<td>$q$</td>
<td>Reference dynamic pressure</td>
</tr>
</tbody>
</table>
\( q \)  
Corrected reference dynamic pressure

\( t \)  
Time

\( u \)  
Inlet velocity

\( u_i \)  
Instantaneous velocity in \( i^{th} \) direction

\( u'_i \)  
Fluctuating velocity in \( i^{th} \) direction

\( U_i \)  
Mean velocity in \( i^{th} \) direction

\( V' \)  
Magnitude of velocity at the edge

\( V_o \)  
Magnitude of reference velocity

\( w \)  
Width of the blockage placed in the wind tunnel

\( W \)  
Width of the wind tunnel

\( X,Y,Z \)  
Cartesian coordinates

\( v_n \)  
Yap correction

\( \alpha \)  
Blending factor

\( \varepsilon \)  
Dissipation of the fluid

\( \phi \)  
Blockage ratio in a two dimensional wind tunnel

\( \mu \)  
Dynamic viscosity

\( v_i \)  
Eddy viscosity

\( \rho \)  
Density of fluid

\( \psi \)  
Blockage correction factor

\( \psi_s \)  
Solid blockage correction factor

\( \psi_w \)  
Wake blockage correction factor
\( \Psi \)  Blockage factor based on vehicle frontal velocity profile

\( \Lambda \)  Shape factor

\( \bar{X} \)  Mesh size
Aerodynamics is the physics of motion of gases. The study of this field had gained significance with the advent of the aircraft. Its primary concern was to determine the flow of air around the body (airplane), predict the forces and moments acting on it and to minimize the aerodynamic drag; thus, increasing the performance of flight. Soon after this, aerodynamics was extended to automobiles, structures, and environmental studies. Of these, aerodynamics applied to automobiles has gained popularity very quickly as a result of the development in the motor industry.

In the Early stages of vehicle research, the main concentration was on design and improvement of the components, aesthetics and spatial manipulation. The desire for higher driving speed vehicles for faster modes of road transportation was the main reason for applying aerodynamics in the field of transportation technology. It was because of the fact that, at high speeds, the performance of the engine also depended on the aerodynamic drag caused by the air flow around the vehicle; thus this new branch of vehicle aerodynamics has emerged. The initial development stressed on reducing the aerodynamic drag. This has fascinated aerodynamicists, vehicle designers, and stylists to bring about novel designs and test various body shapes. This has revolutionized the development of vehicle technology. Further, the field was extended to flow of air through the body and flow process within the machinery.
Many similarities exist between the primary purposes of research in aerodynamics of motor vehicles and that of aircraft, such as lowering aerodynamic drag and balancing of forces and moments for directional stability. However, they differ in many significant aspects. The analysis of aerodynamic effects on automobiles are mostly done experimentally, unlike those on aircrafts that are, to a great extent, proposed theoretically, followed by experimentation on scaled models in wind tunnels. The evolution of the car-shape is the result of aerodynamic developments carried out by car makers in their specific purpose wind tunnels. On the other hand, aerodynamic development of road vehicles involves high expenditure.

The capital required for setting up of testing facilities, such as wind tunnels and climatic tunnels proved to be very high. The experiments conducted were not time effective. The quality and the reliability of the results intended, increased the demand for the quality of the wind tunnel and thereby increased the developmental costs steeply. Results published in literature, when properly applied, would have eliminated wind tunnel testing to some extent; but lower drag coefficients could have been achieved only at the expense of high cost, time, and painstaking process of preparing the full-scale model and then conducting tests on it. Scaled model test results were subjected to numerous doubts associated with realistic simulation of Reynolds number, surface and underbody details, engine cooling and passenger compartment flows, tunnel wall boundary layer and model support interference effects and effects of flow-intrusive probes. On top of all these, the basic and most important wind tunnel experimental
factors that are needed to be taken care of, are the blockage effects. These blockage effects account for drag variation compared to the actual road testing.

In order to overcome some of the problems associated with wind tunnel testing, experts and researchers initiated the development of flow simulations using computers, in the early seventies. Many mathematical techniques and models have been developed so far to provide better and efficient algorithms related to the flow simulations. Computational Fluid Dynamics (CFD) has emerged as one such tool in aerodynamic development of motor vehicles. Recent advances in computing techniques have proved to have reduced the cost and time of analysis, considerably. The following are the advantages of using CFD over wind tunnels:

1. Can be cheap and quick, due to the high computation speeds of modern computer
2. Detailed information in both space and time is obtained
3. Empirical input need not be used
4. No scaling effects are required
5. Analysis can be done on wide range of shapes
6. It can be used for validating the wind tunnel test results

Tremendous research has been done experimentally to determine the blockage effects of the blockage placed in the wind tunnel test section by the Motor Industry Research Association (MIRA), College of Aeronautics (CoA), National Research Council of Canada (NRCC), and many others. The experimental results have been documented by the SAE International. The aim of this present work is, to undertake test cases of simple blockages placed in a virtual wind tunnel test section, created using CAD
modeling software. Initially, a simple two dimensional rectangular block under unconstrained free stream condition is considered and drag behavior curve for different $l/h$ (Length/Height) ratios is determined, in order to gain confidence of the CFD experiments being performed in the present study. The computational results are compared with those of the experimental characteristics and drag correction equation is determined.

The computational study is extended to simple two dimensional and three dimensional blockages subjected to constrained flow inside wind tunnel test sections. The blockage ratio is varied, keeping the same blockage dimensions and increasing the height of the wind tunnel test section. The flow parameter profiles and the drag created by the blockage for different blockage ratios are determined using the CFD codes, AIRFLO3D, and PHOENICS. The blockage correction equations are determined. These blockage correction equations can be applied for other cases and the corrected aerodynamic coefficients can be determined. A grid independent study has been performed to determine the effect of computational grid size on the flow parameters.
In this Chapter, a brief review of the general working and configuration of a simple wind tunnel, the associated blockage effects, and the work done by many researchers in the past related to the study, are provided. The literature review includes the recent advances made in the methods adopted for the determination of blockage effects in wind tunnels using experimental and numerical procedures.

2.1 Wind Tunnel Testing and Blockage Corrections

In this section, the basic functioning of a wind tunnel is discussed. A brief description about the various components and the structure of the wind tunnel is given. Along with this, the fluid mechanics involved in the wind tunnel blockage effects and the role of CFD in determining the blockage corrections, is also described.

2.1.1 An Introduction to Wind Tunnel and its General Layout

A Wind Tunnel is a research tool developed to assist with studying the effects of air moving over, or around solid objects (1). Air is blown or sucked through a duct equipped with a viewing port and instrumentation, where models or geometrical shapes are mounted for study (1). The effect of wind on other vehicles, e.g., automobiles, and on stationary objects, such as buildings and bridges may also be studied in wind tunnels (2). Wind tunnels may be classified based on different categories. But, based on a broader
classification, i.e., according to the type of air guidance, they can be subdivided into closed-circuit and open-circuit wind tunnels. Properties like velocity, pressure, and temperature are also taken care of while constructing a wind tunnel. The quality of the tunnel can be described by the range of the Reynolds number and Mach number that can be tested, along with the turbulence levels and the testing equipment. Quantities generally given are, the maximum speed in the test section, the size of the test section, and the power of the motor (4).

A wind tunnel mainly consists of three components required for basic functioning. These components are: Test section, nozzle and settling chamber, and fan and drive. Flow straighteners, corner vanes, honeycomb layers for reduced turbulence, air exchangers and diffusers are some other features that can be installed for improving the flow conditions. The test section is where, the model is placed and held with appropriate struts. The section is generally rectangular. The longitudinal dimension is about twice the maximum dimension of the section, or, a little more than that (4). The fan is installed depending on its requirement to blow air through the nozzle into the test section, and then to the collector, or to suck air from the nozzle though the test section into the collector. Air flowing through the nozzle into the test section possesses a certain velocity profile and this can be altered using the above mentioned additional features. This air flows over the blockage inserted in the test section, from where pressure, velocity, and other parameters can be measured using appropriate instrumentation. The overall size of a wind tunnel is determined, above all, by the dimensions of the test section. The ratio of the frontal area of the blockage, to the cross-section area of the wind tunnel is called the blockage ratio. It
is desired that the blockage ratio to be as small as possible. The schematic of simple closed circuit wind tunnel and open circuit wind tunnel are shown in the Figures 2.1 and 2.2.

Figure 2.1 A typical Closed Circuit wind tunnel (20)

Figure 2.2 A typical Open Circuit wind tunnel (25)
2.1.2 Wind Tunnel Blockage Corrections

Since the wind tunnel test section is of a confined volume, the aerodynamic measurements obtained from the wind tunnel tests, do not resemble to that of those obtained in infinitely spaced boundaries, such as the case of a vehicle moving on a plane road. Various techniques are used to study the actual airflow around the geometry, and compare it with theoretical results, which must also take into account the Reynolds number and Mach number for the regime of operation (1). One of the most important factors to be taken care of is the boundary corrections at the walls. It is important to have corrections for the boundary effects, if the wind tunnel is to provide a more accurate measure of on-road situation, where, there is no constraint to the flow above or along the sides of the vehicle (3).

Limitation may not be posed on the utility of the wind tunnel as a developmental tool because of the lack of a standard boundary correction procedure. It is because; the differences measured between configurations that are used to guide design, are usually insensitive to boundary interference. But, a proven correction method should lead to an improvement in the absolute accuracy of wind tunnel measurements (3). Moreover, the fuel consumption of a vehicle depends on the drag, to some extent, and the accuracy in the drag measurements can provide accurate performance predictions. A variety of theoretical and empirical methods have been used to adjust wind tunnel data to the unconstrained values; but an examination of these correction procedures indicate that the magnitude of the adjustments provided by them to a given test geometry, may vary by more than a factor of four (3). The cause of this variability is due to the application of the
corrections currently used for automotives, ranging from those based on streamlined aeronautical configurations, to those developed for geometries that have significant regions of flow separation.

2.1.3 Basic Mechanism associated with Blockage Effects

The effect of the blockage can be categorized into three components. Solid blockage, and wake blockage which cause flow speed to increase near the body (blockage), and boundary induced wake related increment in wind axis drag. Solid blockage is the blockage, which is the characteristic of the blockage volume and the wake bubble created next to it. The flow speed in this region of the wind tunnel test section increases relatively with respect to the free stream velocity. The pressure decreases with respect to the initial entry pressure. This can be observed from the plot in Figure 2.3.

![Figure 2.3 Plot for Velocity and Pressure Distributions in a Wind Tunnel Test Section due to the effect of Solid Blockage (3)](image)

Figure 2.3 Plot for Velocity and Pressure Distributions in a Wind Tunnel Test Section due to the effect of Solid Blockage (3)

 Wake blockage is associated with the boundary induced flow acceleration formed due to the developing viscous wake. It is also formed because of the presence of the test
section walls due to the effect of the viscous displacements effect of the wake. This in turn, states that wake blockage is related to wind axis drag. Figure 2.4 provides a clear graphical interpretation of the effect of wake blockage on the variation of velocity and pressure along the wind tunnel test section.

![Figure 2.4 Plot for Velocity and Pressure Distributions in a Wind Tunnel Test Section due to the effect of Wake Blockage (3)](image)

The combined effect of solid and wake blockages components, is shown in the plot, in Figure 2.5.

![Figure 2.5 Plot for Velocity and Pressure Distributions in a Wind Tunnel Test Section due to the combined effect of Solid and Wake Blockage Components (3)](image)
The pressure gradient produced due to the wake source that acts on the model volume, is the reason for the wake-induced drag increment. It has been found that the wake-induced drag increment is proportional to the square of the viscous drag coefficient and independent of pressure gradient or the volume of the model.

2.1.4 Role of CFD in Wind Tunnel Blockage Corrections

When Computational methods were applied, the testing process became easier and faster. CFD, a flow analysis tool, allowed the usage of larger models in a wind tunnel, which reduced experimental error and problems associated with scaling (16). The drawbacks, such as loss of coefficient of free jet and the unimpeded sound radiation were completely eliminated. Wind tunnel testing had been a very tedious process. The aerodynamic measurements obtained in a wind tunnel were appropriate only to that particular wind tunnel. The test results differed from the results of some other wind tunnel for the same bluff body. If the obstruction had been more appreciable in one tunnel compared to the other and in such cases, the data was not comparable.

Earlier, Computational Fluid Dynamics (CFD) methods were used to simulate for conditions under free stream. This was the easiest way for analyzing air flow around bodies numerically. But these computational results, now, are generally used to compare the experimental data, which may be biased by wall temperature. Either of the results, computational, as well as the experimental, needs to be corrected. Though this involves additional cost and time, the comparison of the results could lead to a more accurate analysis of the flow of air over various blockages in a wind tunnel.
Numerical modeling has so far been done by adopting many techniques, and in combination of many types of software which involve solid modeling, grid generation, solving, and post-processing the results. Though the computational results cannot completely replace the experimental results, attempts have been made by many researchers to attain accuracy to maximum extent. For this, experimentations have been carried out on simple wind tunnel blockages, and varying wall interference conditions. Many researchers have successfully generated computer codes for increasing computational efficiency, reducing computational cost and time, and maintaining stability of the computational techniques used. A huge content of documented works has been provided, out of which relevant information will be discussed in the literature review.

2.2 Literature Review

Till now, significant work has been done, to have a better understanding about the blockage corrections applied to wind tunnels. Research has been done to determine blockage correction equations both experimentally and computationally. This section provides a brief review of some contributions done in this research field, so far.

2.2.1 Review of Work on Experimental determination of Blockage Correction Equation

Maskell (6), in his experiments, derived correction equations applied to flat plates stalled at the test section centers. His theory is based on the principle of momentum balance. His procedure did not take into account, the solid blockage effects; but only those effects caused due to flow separation. However, the base pressure was not known
under zero-blockage, and was assumed to be uniform. The blockage correction, he proposed for bluff bodies, might be looked up at as a velocity increment. It was because, there would be no drag increment present, as a flat plate has no volume. Maskell’s correction is based on the assumption that,

$$\frac{C_D}{C_{Dc}} = q$$  \hspace{1cm} (2.1)

The blockage equation developed by Makell to determine the blockage correction for thin plates is,

$$\psi = \frac{C_D}{C_{Dc}} = 1 + \phi \frac{A_m}{A_w}$$  \hspace{1cm} (2.2)

Thom (6), has provided the estimation of blockage correction for streamlined and attached-flow bodies in closed-test-section wind tunnels of various cross-sectional shapes. While, Herriot (8) conducted the same experimentation to cover a range of aeronautical body and wing shapes, and provided with tabulation of the appropriate constants. Thom and Herriot suggested that an addictive wake-induced incremental drag adjustment was to be applied before the dynamic pressure correction (3).

Ramamurthy and Ng (9), have conducted wind tunnel experiments with an object placed in series, and found that the drag was same as that obtained for a single object at identical blockage levels. In their experiments, Ramamurthy and Ng have proposed blockage correction equations for two-dimensional rectangular and triangular prisms for different blockage ratios. This has facilitated the evaluation of interference effects of a number of objects placed in series, just with experiments conducted on a single object.

Ranga Raju and Vijaya Singh (5), based on the works done by Maskell (6), Ramamurthy
and Ng (9) from the experiments that they conducted, found that the base pressure was independent of plate height, plate thickness, and the Reynolds Number, if it is greater than \(10^3\). But, it varied with varying blockage ratios. They proposed blockage correction equation for objects submerged in boundary layer.

Awbi (10), showed that the effect of confinement of the mean surface pressure on the model not to be uniform in the stream wise direction, and could not be regarded as an increase in the effective dynamic pressure estimated from the drag, or base pressure measurements. He came up with a new correction equation which is a modification of Maskell’s equation and this included a shape factor to allow for the depth of rectangular section, satisfactorily yielding corrected drag and base pressure. The shape factor was included in order to differentiate between the flow over the bluff body under consideration, and that over a flat plate (11). He used \(K_D\), replacing \(\phi\) in Maskell’s equation, equation (2.2).

\[
K_D = \phi \Delta \\
\Delta = 1.11 + 0.94\left(\frac{l}{h}\right), \quad \text{for} \quad 0 < \frac{l}{h} < 0.5 \tag{2.3}
\]

\[
\Delta = 1.11 - 0.14\left(\frac{l}{h}\right), \quad \text{for} \quad 1 < \frac{l}{h} < 5 \tag{2.4}
\]

This has attributed to, establishing a rational approach for extending the theory to a wide range of bluff bodies. His results were in good agreement with those of Maskell and Glauert (12). He observed that the drag coefficient had a maximum value at a critical \(l/h\) ratio of 0.62, for the rectangular section, for a given \(h/H\) ratio, where, \(l\) is the width of blockage, \(h\) is the height of blockage, and \(H\) is the height of the wind tunnel. It was also observed by him, that the maximum \(C_d\) increased with increasing \(h/H\) ratio.
Merek (13), had developed blockage correction for automotive blockages. His correction procedures provided a distinction between solid and wake blockages. His solid blockage factor was based on the work of C.N.H. Lock (14), who derived solid blockage correction taking into account, the shape factor of the object being tested. The wake blockage factor he derived, was based on the works of Maskell, Thom, and Glauert. In his work, he expressed blockage correction in the form of dynamic pressure, as shown,

\[ q_c = q_0 (1 + \psi)^2. \]  
\[ \text{(2.6)} \]

Or

\[ q_c = q_0 (1 + 2\psi). \]  
\[ \text{(2.7)} \]

The above equation has been modified, to obtain the correction in terms of solid and wake blockages, which is as follows,

\[ q_c = q_0 (1 + 2\psi_s + 2\psi_w). \]  
\[ \text{(2.8)} \]

He performed wind tunnel experiments on the automobile shapes that were standardized by the MIRA (Motor Industry Research Association), and applied regression analysis on the aerodynamic measurements. From these experiments, he also proved that the blockage correction equations are valid up to, 30° of yaw angle of the object placed in the wind tunnel.

Jeffrey et al. (22) have performed experiments on the MIRA models in the Drivability Test Facility (DTF) wind tunnel, for 7%, 11%, 15%, 20% and 25% blockage ratios, and have developed correction equations for drag and lift for open jet wind tunnels. From their experimental results, they observed that vehicle location and blockage ratio pose a significant effect on the aerodynamic drag interference errors. They also
predicted that the interference error in lift measurements, increased with increase in the
vehicle size.

2.2.2 Review of Work on Computational determination of
Blockage Correction Equation

Louis (14), in his paper, presented a numerical method, by which the walls of a
closed-working-section wind tunnel could be so shaped, that the blockage effects for a
particular body could be considerably reduced. He compared his results with the
experimental work done on the same blockage in a large, low blockage wind tunnel. The
results were found to be in good agreement. These numerical solutions and experimental
results produced from streamline tunnels, were, better (less blockage effects) compared to
these obtained using parallel-walled tunnel. Using the mathematical model, the optimum
wall geometry was also determined.

Atsushi et al. (15) presented a numerical simulation for flow around stationary
and oscillating rectangular cylinders for various blockage ratios, to study the blockage
effects on the aerodynamic characteristics of the cylinder. Using Direct Simulation (DS)
method for laminar, and k-ε model for turbulent flows, they obtained corrected pressure
and velocities, applying Semi-Implicit Method for Pressure-Linked Equations (SIMPLE)
Algorithm for solving the Reynolds Averaged Navier-Stokes (RANS) equation. Their
computational results were in good agreement with the experimental results. Their
simulation observations showed that the drag and lift forces, and the Strouhal number, all
increase with increasing blockage ratios.
Lasher (16), in his work, proposed a CFD method to determine the blockage effects on two-dimensional flat rectangular plates. He considered velocity components in $x$ and $y$ directions as 1.0 and 0.0, respectively, producing a Reynolds number equivalent to 32200, and exhibiting turbulence flow condition. He adopted a finite-element based, commercial CFD code called FIDAP for his numerical experimentation. His results for quasi-steady state, showed that there was no significant difference when various turbulence models were used for predicting $C_d$. He observed that the difference between the experimental and computational drag was constant for different blockage ratios. His drag coefficient results were under-predicted, otherwise compared to those obtained from many other turbulent models. His results, also showed that problem domain size and mesh distribution, play a very vital role in determining the accuracy of the solution. This work represented the use of quasi-steady simulation as a design tool in determining blockage corrections.

A much more advanced research was done by Sven (17) at the Volvo Car Corporation. He came up with an idea of simulating a representative solid wall wind tunnel, instead of a slotted wall wind tunnel, as it was a very painstaking job to simulate such a tunnel, and because of the uncompromisingly exorbitant computational costs. His work was not exactly about blockage factors, instead, about the comparisons between two different sets of CFD data, covering the PVT and on-road or free-stream conditions. Like many other computational experts, he utilized the symmetric wall condition and simulated only, half the domain and blockage geometries, which reduced the computational cost and time, considerably. He worked on both two dimensional and three
dimensional simulations. His two dimensional analysis was done as a parametric study of wind tunnel blockage against various blockage geometric variables. Using blending technique, like the weighting coordinates in every point, he made it possible, to test numerous amounts of configurations with less meshing effort. The equation he used is,

\[
\bar{\chi} = \alpha \frac{\chi}{\text{mesh 1}} + (1-\alpha) \frac{\chi}{\text{mesh 2}}.
\]

(2.9)

For his three dimensional simulation, he used a research geometry, called Volvo Research Aerodynamic Knowledge (VRAK) car body, second-order discretization scheme called MARS, standard k-ε turbulence model, wall functions, and StarCD solver. His results showed that, in case of two dimensional simulations, none of the parameters tested, were significant for low blockage ratios; i.e., for blockage ratios less than 5%. However, drag improvements are possible only at low blockage ratios. On the other hand, the results from the three dimensional simulations, indicated a variation in wake structure for Volvo wind tunnel and free-stream conditions. Finally, he inferred that, “what’s working for one geometry may not work for another” (17).

One of the most recent works on CFD study on blockage corrections in wind tunnels was done by Yen et al. (18). They have modeled different blockage effects in a Climatic Wind Tunnel (CWT) using four basic vehicle shapes, a sedan, SUV, pickup truck, and a minivan. Their approach was to take into account, two different nozzle-cross-sectional areas, which differed due to the change in the nozzle height. For all the flow simulations, they used Star-CD v.3150, a commercial CFD package from Computational Dynamics, Inc. The flow solution domain was discretized by creating a hexahedral mesh using ICEMCFD Engineering, Hexa software. The simulation matrix was solved using
the same MARS scheme and Suga nonlinear turbulence model. They considered domain sizes on the CFD models varying from, $0.7 \times 10^6$ cells for an on-road simulation to, $3.4 \times 10^6$ cells for a CWT simulation. In the case of on-road situation, blockage ratio of 2% was considered, in order to reduce false (numerical) blockage effects. For determining the velocity corrections, they used the simulated Rout’s (19) experimental method of correlating the vehicle frontal velocity profiles, of a test data gathered from on-road situation and CWT simulation. This approach resulted in a specific velocity correction method for tests done in CWT. From this unified correlation and by applying linear regression, Yen et al., obtained blockage factor of the form,

$$ \psi_{v,\text{FrontalVelocity}} = 0.9412 C_d \sqrt{A} / D_{WB} + 0.9129. \quad (2.10) $$

Regarding blockage corrections by vehicle upper surface centerline pressure traces, they used a method described by Hucho (20). Hucho determined experimentally, blockage correction factor by plotting surface static pressure coefficient measurements from wind tunnel test, against those obtained from road tests. Applying boundary layer theory, they corrected the surface pressure trace to an effective velocity trace, along the vehicle surface. This was obtained by using the expression for edge surface velocity magnitude.

$$ \frac{V_e}{V_o} = \sqrt{1 - C_{pe}}. \quad (2.11) $$

From the results obtained from the linear regression of the surface pressure traces, for both test cases, they obtained the blockage correction factor,

$$ \psi_{v,\text{SurfacePressure}} = 0.6740 C_d \sqrt{A} / D_{WB} + 0.9623. \quad (2.12) $$
Hence, their study had revealed a different blockage effect in the flow field of an automobile vary with vehicle body contour, geometry blockage ratios, and the blockage position from the nozzle exit point (NEP).

At General Motors Corporation, Yang et al. (21), in their work, stated that the variation of the blockage effect is attributed to the change in wind tunnel height. They found that, due to smaller wind tunnel heights, the blockage increased for some vehicle models and thereby increased drag. As found from their CFD study, the sum of the corrections and the residual was not, in all cases, equal to the difference in drag coefficient due to round-off errors. By changing the cross-sectional area of the virtual wind tunnel (CFD simulation), the pressure gradient effects for a given experimental wind tunnel were captured by CFD analysis.

\[ \Delta C_d(Expt) = \Delta C_d(CFD). \]  \hspace{1cm} (2.13)

\[ C_d = C_d(Expt) - \Delta C_d(CFD). \]  \hspace{1cm} (2.14)
CHAPTER 3
COMPUTATIONAL METHODOLOGY

The main focus of this study is to determine the blockage corrections for different blockages placed in the wind tunnel test section, using CFD methods and schemes. This chapter mainly concentrated on the methodology followed in achieving the final goal of this thesis. A detailed review of the various assumptions, tools, and methods used in this research work is given in this chapter.

This study deals with numerical simulation of four cases of flow over blockages. These four cases are:

A. Prediction of drag coefficient variation for airflow over a two dimensional rectangular blockage, under free stream condition and varying \( l/h \) ratio and comparing the results with the experimental data.

B. Prediction of drag coefficient variation for airflow over a two dimensional rectangular blockage placed in a virtual wind tunnel under varying blockage ratio, and to determine the blockage correction equation.

C. Prediction of drag coefficient variation for airflow over a three dimensional rectangular blockage placed on the floor of a wind tunnel, and to determine the blockage correction equation.

D. Prediction of drag coefficient variation for airflow over a three dimensional top-front edge tapered rectangular block, placed at a little height above the floor, and to determine the blockage correction equation.
3.1 Assumptions

While following the methodology, many assumptions have been taken into account. These assumptions that are considered for running the numerical simulations are that, in the flow analysis, temperature variations and heat transfer are neglected. Compressibility effects are not accounted for. Blockages are tested only at zero angle of attack. The effects of gravity are not considered. Flow over blockages under steady state only, is taken into consideration. A uniform profile Inlet airflow velocity of 1m/sec is assumed in all the cases, irrespective of the variation of the Reynolds number. While determining the drag coefficient, the effect of only the pressure drag is considered.

3.2 Computational Setup and Solution Procedure

The details of the test section dimensions, blockage dimensions and their location in the testing domain are discussed. The modeling of the test section domains and the blockages, is done using Pro/E Wildfire of the PTC Inc. The modeling is done taking into account, all the dimensions in millimeters.

3.2.1 Domain Layout Considerations

In order to reduce the difficulties in the analysis two dimensional flow cases, three dimensional models are created, in which the third dimension, which is the projected thickness of the two dimensional sketch, has no effect on the two dimensional flow analysis for drag, according to Ramamurthy and Ng (9). The blockages used in the wind
tunnel test section are rectangular, in case of two dimensional testing. The blockage ratio in the two dimensional case is defined as the ratio of the height of the blockage to the height of the test section or the domain.

\[ \phi = \frac{h}{H} \quad (3.1) \]

For the free stream flow condition, 1\% blockage ratio is considered so that, the effect of the walls on the flow over the blockage is very negligible. The dimensions of the computational setup are provided in detail, in the Figures 3.1 and 3.2.

Figure 3.1 Layout of the free stream domain with blockage; height of the blockage, \( h = 400 \text{mm} \)

In the above case, the \( l/h \) ratio is varied and hence the length of the blockage changes with it. The following table, Table 3.1, illustrates the variation.
Table 3.1 Magnitude of \( l \) for different \( l/h \) ratios

<table>
<thead>
<tr>
<th>( l/h ) Ratio</th>
<th>0.7</th>
<th>1.2</th>
<th>2.0</th>
<th>2.5</th>
<th>3.0</th>
<th>6.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>( l ) in mm</td>
<td>280</td>
<td>480</td>
<td>800</td>
<td>1000</td>
<td>1200</td>
<td>2400</td>
</tr>
</tbody>
</table>

Figure 3.2 Layout of the wind tunnel test section with the rectangular blockage, \( h=400\text{mm} \)

The rectangular blockage has constant dimensions. So, the height of the wind tunnel is decreased with increasing blockage ratio. The blockage is seen to that, it is always located in the midway of the total height of the test section. Table 3.2 depicts magnitudes of the height of the test section and the vertical location of the blockage, with varying blockage ratio.

Table 3.2 Height of the wind tunnel and the vertical location of the blockage for 5\%, 10\% and 15\% blockage ratios

<table>
<thead>
<tr>
<th>Blockage Ratio %</th>
<th>5</th>
<th>10</th>
<th>15</th>
</tr>
</thead>
<tbody>
<tr>
<td>( H ) (x*( h )) mm</td>
<td>6800 (17( h ))</td>
<td>3600 (9( h ))</td>
<td>2800 (7( h ))</td>
</tr>
<tr>
<td>( K ) (x*( h )) mm</td>
<td>3200 (8( h ))</td>
<td>1600 (4( h ))</td>
<td>1200 (3( h ))</td>
</tr>
</tbody>
</table>
The three-dimensional model created, is a rectangular solid-wall wind tunnel test section. The test section and the blockage dimensions have been taken with reference to the automobile model dimensions suggested by the MIRA and experimental data provided by the SAE in the reference (3). The blockage ratio in the three-dimensional flow analysis is defined as the ratio of the projected cross sectional area of the blockage, to the cross sectional area of the wind tunnel test section. This is denoted by the equation,

$$\phi_{bd} = \frac{A_{\text{Blockage}}}{A_{\text{Test Section}}} = \frac{w \cdot h}{W \cdot H}.$$  

(3.2)

The height of the test section in the three dimensional case is also varied with change in the blockage ratio. So, the initial height of the test section and the dimensions of the blockages are taken as standard, at 15% blockage ratio. Then, keeping the dimensions of the blockage constant, the height of the wind tunnel test section is increased proportionally, for 10% and 5% blockage ratios.

In the present research, the blockage geometries considered are rectangular cuboid and a rectangular block with top-front edge tapered. Both the geometries are created to have their maximum projected dimensions to be equal. The simple rectangular blockage is positioned on the floor of the wind tunnel test section. Whereas, the blockage with top front edge tapered, is positioned at a certain height above the floor. In both the cases, the positions of the blockages with respect to the floor are unaltered, irrespective of the changes in the test section height. The actual layout of the blockages and their location in the wind tunnel test section, are as shown in the Figure 3.3 and 3.4.
Figure 3.3 Orthographic views of the Wind tunnel and the rectangular blockage for three dimensional flow analysis

Figure 3.4 Orthographic views of the wind tunnel and the top front edge tapered rectangular blockage, for three dimensional flow analysis

In the above two cases, the test section dimensions and the blockage dimensions are similar. So the variation in height of the test section roof with respect to the blockage ratio is shown in Table 3.3.
Table 3.3 Test section height variation with blockage ratio

<table>
<thead>
<tr>
<th>Blockage ratio %</th>
<th>5</th>
<th>10</th>
<th>15</th>
</tr>
</thead>
<tbody>
<tr>
<td>H in mm</td>
<td>3086.95</td>
<td>1543.48</td>
<td>1029.00</td>
</tr>
</tbody>
</table>

If observed, the three dimensional analysis is done only on half the wind tunnel and the blockage geometries. It is done in order to reduce the computational time and computational costs. This is incorporated by assigning the symmetric wall condition at the plane of symmetry. The plane of symmetry for the two cases lies perpendicular to the floor, along the longitudinal axis of the wind tunnel at the mid way, and is represented by broken line. For convenience of notation, the four cases are assigned (A), (B), (C) and (D), respectively.

3.2.2 Mesh Generation

In order to comply with the limitations of the mesh generation packages for the flow analysis, the modeling and interface generation in Pro/E is done in two ways. (a) The wind tunnel test sections and the free stream domains are created as finite protrusions and the blockages are created as cut-outs of the finite projections. These completed models are then transformed into ASCII format with extension *.iges and exported to the Indigo Silicon Graphics System at the CFD Lab at Texas Tech University, for mesh generation in ICEM CFD Engineering v.4.1.
The blockages are created as finite protrusions and then transformed into ASCII format with extension *.stl and exported to PHOENICS v.3.5 flow analysis software for mesh generation.

The ICEM CFD Engineering package is exclusive mesh generation software developed by ANSYS, Inc. It is a meshing tool developed for CFD purposes, as CFD applications require computational domain for analysis. ICEM CFD possesses the ability to generate a wide variety of mesh elements. In the present work, the mesh generation is done with hexahedral grid using ICEM CFD Hexa tool. This package also facilitates assigning of the boundary conditions to the walls of the computational domain and the blockage.

PHOENICS is flow analysis software developed by CHAM Ltd. The domain creation, mesh generation, assignment of the boundary conditions, flow model, solver end time and the number of iterations, is done in the pre-processor called VR Editor. Due to the limitations of the package, symmetric wall conditions were not applied for analysis done in PHOENICS. So, the analysis had to be done on the whole model in cases (C) and (D). For cases (A) and (B), the thickness of the domain has been taken as 10h in order to comply with the symmetric wall condition.

The mesh is generated with uniform cell spacing at the vicinity of the blockage. But, a geometrically increasing cell size is considered, as the cell position moves away from the blockage. This also saves computational time and cost. The mesh generated using ICEM CFD for the four cases, taking a single blockage of each case, are shown in the Figures 3.5, 3.6, 3.7 and 3.8. The mesh generated using the VR Editor in PHOENICS,
are shown in Figures 3.9, 3.10, 3.11 and 3.12. The mesh has been generated for various blockages using Blending Technique, by which the mesh sizes of intermediate blockages are determined using the equation (2.9). This method helps achieve a smooth transition of the mesh sizes for test domains from lower blockage ratios to higher and vice versa. In order for the blending technique to be implemented, the mesh sizes of the two blockage ratios are to be known. The blending factor for the case (A) are taken as, 0.8, 0.4, 0.2 and 0.1. Taking the mesh1 and mesh2 as the first and last mesh sizes, the intermediate mesh sizes are determined using the above blending factors, respectively. For the remaining cases, the blending factor is taken as 0.8.
Figure 3.5 Diagram of the mesh generated in ICEM CFD for the case (A). (a) Isometric View. (b) Side View
Figure 3.6 Diagram of the Mesh generated in ICEM CFD for the case (B). (a) Isometric View. (b) Side View
Figure 3.7 Diagram of the mesh generated in ICEM CFD for the case (C). (a) Isometric View. (b) Side View
Figure 3.8 Diagram of the mesh generated in ICEM CFD for the case (D). (a) Isometric View. (b) Side View
Figure 3.9 Diagram of the mesh generated in PHOENICS VR Editor, for the case (A).
(a) Side View. (b) Front View
Figure 3.10 Diagram of the mesh generated in PHOENICS VR Editor, for the case (B).
(a) Side View. (b) Front View
Figure 3.11 Diagram of the mesh generated in PHOENICS VR Editor, for the case (C).
(a) Side View. (b) Front View
Figure 3.12 Diagram of the mesh generated in PHOENICS VR Editor, for the case (D).
(a) Side View. (b) Front View
The mesh sizes of the domains of all the four cases are tabulated in Table 3.4 and 3.5, as follows.

### Table 3.4 The mesh sizes of the computational domain for case (A), taking into account the blending factor $\alpha$

<table>
<thead>
<tr>
<th>Blending factor $\alpha$</th>
<th>Mesh sizes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.7</td>
</tr>
<tr>
<td>$l/h$</td>
<td></td>
</tr>
<tr>
<td>Case (A) ICEMCFD</td>
<td>45700</td>
</tr>
<tr>
<td>Case (A) PHOENICS</td>
<td>56280</td>
</tr>
</tbody>
</table>

### Table 3.5 The mesh sizes of the computational domain for cases (B), (C) and (D), taking into account the blending factor $\alpha$

<table>
<thead>
<tr>
<th>Blending factor $\alpha$</th>
<th>Blockage ratio%</th>
<th>Mesh sizes</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>5</td>
<td>10</td>
</tr>
<tr>
<td>Case (B) ICEMCFD</td>
<td>46980</td>
<td>37680</td>
</tr>
<tr>
<td>Case (B) PHOENICS</td>
<td>111300</td>
<td>84820</td>
</tr>
<tr>
<td>Case (C) ICEMCFD</td>
<td>81400</td>
<td>65470</td>
</tr>
<tr>
<td>Case (C) PHOENICS</td>
<td>156000</td>
<td>109200</td>
</tr>
<tr>
<td>Case (D) ICEMCFD</td>
<td>54140</td>
<td>41040</td>
</tr>
<tr>
<td>Case (D) PHOENICS</td>
<td>106580</td>
<td>82510</td>
</tr>
</tbody>
</table>
3.2.3 Boundary and Initial Conditions

Boundary conditions are assigned to the walls of the computational domains after the mesh has been generated. The assignment of the boundary conditions to the test domains differs between ICEM CFD and VR Editor, because symmetric plane assignment feature is not available in PHOENICS. It requires a separate sub-routine to be written in C or FORTRAN to incorporate the feature. The boundary conditions assigned are as shown in Figures 3.13, 3.14, 3.15, 3.16, 3.17, 3.18 and 3.19.

Figure 3.13 Boundary conditions used in ICEM CFD for flow in case (A). (a) Side View. (b) Front View
Figure 3.14 Boundary conditions used in ICEM CFD for flow in case (B). (a) Side View. (b) Front View
Figure 3.15 Boundary conditions used in PHOENICS for flow in cases (A) and (B). (a) Side View. (b) Front View
Figure 3.16 Boundary conditions used in ICEM CFD for flow in case (C). (a) Side View. (b) Front View
Figure 3.17 Boundary conditions used in PHOENICS for flow in case (C). (a) Side View. (b) Front View
Figure 3.18 Boundary conditions used in ICEM CFD for flow in case (D). (a) Side View. (b) Front View
Figure 3.19 Boundary conditions used in PHOENICS for flow in case (D). (a) Side View. (b) Front View
3.2.4 Turbulence Models and Numerical Methods

In most of the engineering flow application, the flow is turbulent, and so it is, in the current study. It is essential to have a brief knowledge about turbulence and the models that are used in this work. Turbulence is an irregular flow structure in which the flow quantities, such as velocity and pressure randomly vary with time and position. In spite of the irregularity, turbulence obeys the conservation laws. Rotational flow structure or the formation of eddies is a predominant characteristic of turbulence. Due to the dominance of viscous stress, eddies die down, which results in the loss of kinetic energy of the fluid.

The most general equation used for solving a turbulence flow is the time dependent, three dimensional Navier Stokes Equation, shown below

\[
\rho \frac{Du_i}{Dt} = \frac{\partial P}{\partial X_i} + \frac{\partial}{\partial X_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right].
\]  
(3.3)

Where \( u_i \) is the instantaneous velocity and is expressed as,

\[
u_i = U_i + u'_i.
\]  
(3.4)

The above equation, equation (3.4) is the statistical representation of an averaged turbulence quantity, as a sum of mean and fluctuating terms.

From equations (3.3) and (3.4), the Navier Stokes equation takes the form,

\[
\rho \frac{DU_i}{Dt} = \frac{\partial P}{\partial X_i} + \frac{\partial}{\partial X_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho u'_i u'_j \right].
\]  
(3.5)

The above equation is often called the Reynolds Averaged Navier Stokes (RANS) equation. The term \( \rho u'_i u'_j \) is an unknown, and on comparing the terms of the equation, it
was found to be turbulence stresses. These stresses were later called Reynolds stresses, named after the person who proposed the equation. Due to the presence of the stress term, the RANS equation is not closed. In order to obtain a closure problem for the above equation, several models have been proposed to express the stress in terms of dependent variables. The turbulence model used in this study is the K-\(\varepsilon\) model, proposed by Launder and Spalding (1974) (23). The model solves the turbulence flow problems with just the initial and the boundary conditions specified and without any prior knowledge of the turbulence structure.

The governing equations of the K-\(\varepsilon\) model are,

\[
\frac{DK}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + P_k - \varepsilon. \tag{3.6}
\]

\[
\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{K} \left( C_1 P - C_2 \varepsilon \right). \tag{3.7}
\]

The eddy viscosity is given by,

\[
\nu_t = C_\mu \frac{K^2}{\varepsilon}. \tag{3.7}
\]

The production of the turbulence kinetic energy is given by,

\[
P_k = \nu_t S^2. \tag{3.8}
\]

The empirical constants \(C_1=1.44\), \(C_2=1.92\), \(C_\mu=0.09\) \(\alpha_k=1.0\) and \(\sigma_\varepsilon=1.3\), have been applied to the K-\(\varepsilon\) equations and successful results were obtained.

But the main drawback of the K-\(\varepsilon\) model is that, it over predicts the turbulence kinetic energy in the vicinity of the stagnation point, which leads to large eddy viscosity.

It is because, depending on the flow conditions, the production term has to be positive,
zero or negative and with the standard K-\varepsilon model, only a positive value is yielded. So, a modified K-\varepsilon model was proposed by Kato and Launder to avoid this over prediction. The production term is defined by the equation,

\[ P_i = \nu_i S \Omega. \] 

Where \( S \) and \( \Omega \) are the strain rate scale and the vorticity scale defined by the equations,

\[ S = \sqrt{\frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)^2}, \]

\[ \Omega = \sqrt{\frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right)^2}. \] (3.10)

The AIRFLO3D solver, a CFD code developed in by S. Parameswaran, has been used to analyze the flow in this thesis. The modified K-\varepsilon model was incorporated in the code for turbulence modeling. The numerical method incorporates non-linear equations linearized by time, a fully implicit differencing scheme to approximate the temporal terms, a hybrid differencing scheme associated with the Central Differencing Scheme (CDS) for spacial solving. At each time step, the coupled linear equations for velocity components and the pressure are solved by a modified SIMPLE algorithm. The pressure correction equations are subjected to preconditioned conjugate gradient method, while the velocities are solved by Stone’s Strongly Implicit Procedure (SSIP). In the current study, which requires results for steady state, the solution is obtained by marching in time, until the solution remains unchanged. The AIRFLO3D code was run on a supercomputer called Pleione located in the High Performance Computing Center (HPCC) at the Reese Technological Center, Texas Tech University. The Pleione is an
SGI Onyx2 system with 56-300 MHz processors, and 56 GB of shared RAM. The
programming language used is the FORTRAN 90 GL.

The PHOENICS uses a solver called EARTH, to solve the equations over the
domain, with the assigned boundary conditions. The turbulence model chosen is the low-
Reynolds number K-ε model, proposed by Lam Bremhorst. At low Reynolds numbers, in
this K-ε model, the C₁, C₂ and C₃ are modified to be damping functions, Reₙ and Reᵣ, and
are defined by the following equations.

\[
C'_\mu = 0.09 \left[ 1 - e^{-0.0165Re_\mu} \right] \left( 1 + \frac{20.5}{Re_\mu} \right),
\]
\[
C_1 = 1.44 \left[ 1 + \left( \frac{0.0045}{C_\mu} \right)^3 \right],
\]
\[
C_2 = 1.92 \left[ 1 - e^{-Re_\mu} \right]
\]

Where.

\[ Re_\mu = \frac{K^2 \nu}{\nu} \quad \text{and} \quad Re_\epsilon = \frac{K^2}{\epsilon \nu}. \]  

(3.12)

The boundary conditions used are K=0 and \( \frac{d\epsilon}{dy} = 0 \). At high Reynolds numbers, C₁, C₂
and C₃ tend to become constants.

The PHOENICS Earth solver uses an algorithm called X-cell developed from the
Conservative Low-Dispersion algorithm (24). Upwind differencing scheme was used to
solve the continuity, and the momentum equations for convection. The pressure and the
velocities are solved using the SIMPLE algorithm, whereafter, it is ensured to satisfy the
continuity equations.
The solutions obtained from the solvers are post-processed to interpret the data into useful graphical formats. The data is also used for calculating the drag coefficients for the four cases. The definition for drag coefficient used in both the cases is same and is represented by the following equation.

\[ C_D = \frac{F_D}{\frac{1}{2} \rho A u^2} \]  

Where, \( F_D \) is the drag force acting on the body due to the pressure acting on the body, produced as a result of the flow, and \( V \) is the velocity of air in the direction of flow. A detailed analysis of the data obtained from the solvers will be present in the next chapter.
CHAPTER 4

RESULTS AND DISCUSSIONS

The computational methodology in the chapter 3 is used to determine the flow analysis for the four cases mentioned in the same chapter. The results obtained from this computational analysis are compared and blockage correction equations are proposed. The computational results obtained from case (A) are compared with those in the reference (26). Due to unavailability of the wind tunnel experimental results for cases (B), (C) and (D), the results obtained from PHOENICS are used to provide the blockage correction equations for the results obtained from AIRFLO3D.

The data obtained from the AIRFLO3D and the PHOENICS Earth solvers are analyzed after Post-processing in ENSIGHT and GUI Post-processor in PHOENICS, respectively. The number of iterations used for each case is 2000. A detailed description of the results obtained from each test case is presented in this chapter.

The results in each case include the pressure signatures and velocity profiles plotted along the line passing the longitudinal dimension (i.e., the direction of flow) of the domain and touching the upper surface of the blockages placed in the domain. For the cases (C) and (D), the data sets for the plots are obtained about the longitudinal plane of symmetry. The other plots include the drag coefficient and drag correction plots. The following sections deal the results case by case. The pressure signature is the pressure coefficient, which is determined by the equation (4.1).
4.1 Case (A) Two Dimensional Flow around a rectangular section subjected to an unconstrained flow field

The computational domain, with the blockage, as shown in Figure 3.1 and the grid structure as in Figure 3.5, indicate clearly that the computational domain is symmetric along a horizontal line passing through the middle of the rectangular blockage. Since a vertically symmetric velocity profile (constant velocity of 1000 mm/sec) has been applied, the flow pattern also has to be symmetric, vertically. Results have been obtained both in PHOENICS and AIRFLO3D, only along the upper horizontal line of the rectangular blockage, due to the symmetric flow condition, i.e., along the line AB in Figure 4.1.

Figure 4.1 Line along which the Pressure and Velocity Distributions are obtained in Two Dimensional case
It can be observed from the pressure signature plot of Figure 4.2, that the pressure away from the blockage is not effected. It remains almost constant over a considerable distance from the test domain entrance. There is a sudden increase in the pressure at a little distance in front of the blockage; but it decreases as it approaches the edge of the blockage. Past the blockage, there is further decrease in the pressure and reaches below zero pressure. With a little increase past zero pressure, the pressure past the blockage tends to be constant till the exit of the test domain.

The velocity also remains constant at the entrance of the test domain to almost up to the front end of the rectangular blockage. It decreases further, past the blockage and with a slight increase, again tends to remain constant till the domain exit. It can also found that the velocity profile has a very little variation with the change in the $l/h$ ratio, in the vicinity of the blockage. The variations can be clearly observed in the velocity profile plot from Figure 4.3.
Plot for Pressure Signature at the top/bottom line of a Two Dimensional Rectangular Block, not subjected to constrained flow, using the data obtained from PHOENICS

Figure 4.2 Plot for Pressure Signature using PHOENICS data for Case (A)
Plot for Velocity Profile at the top/bottom line of a Two Dimensional Rectangular Block, not subjected to constrained flow, using the data obtained from PHOENICS

Figure 4.3 Plot for Velocity Profile using PHOENICS data for Case (A)
The sudden rise in the pressure can be attributed to the high pressure, due to stagnation in front of the blockage and the change in the velocity direction. The inconsiderable variation in the velocity can be attributed to the reason that, the wall conditions are too far from the blockage to affect. The sudden drop in the velocity beyond the blockage is due to the formation of wake bubble at the rear of the blockage. The effect of wake extends till the domain exit. This might possibly be the reason for the difference in the inlet and outlet velocity magnitudes. The same reason holds for the sudden drop and rise of pressure beyond the blockage. It can also be seen that the wake does not affect the flow much, above or below the blockage.

The plot for variation of drag coefficient with varying $l/h$ ratio is also plotted. The drag plot obtained from PHOENICS data shows a gradual decrease in the drag coefficient with the increase in the $l/h$ ratio. The variation is very little compared to the experimental drag coefficient, plotted against $l/h$. AIRFLO3D yields higher results compared to, PHOENICS and experimental data. It also shows a considerable drag variation. Figures 4.4, 4.5, and 4.6 show the drag coefficient variations for the data from the three sources.

The drag correction for the computational results can be obtained from a plot drawn for computational drag against experimental drag. Linear regression is applied to the data points in the plot and the linear equation obtained from it is the blockage correction equation. The plots for the blockage correction can be seen in Figures 4.7 and 4.8.
Plot for Drag Coefficient variation with l/h ratio of a two dimensional block, not subjected to constrained flow conditions, using Experimental data

Figure 4.4 Plot for Drag Coefficient variation with l/h ratio using Experimental data for Case (A)
Plot for Drag Coefficient variation with l/h ratio of a two dimensional block, not subjected to constrained flow conditions, using PHOENICS data

Figure 4.5 Plot for Drag coefficient variation with l/h ratio using PHOENICS data for Case (A)
Plot for Drag Coefficient variation with l/h ratio of a two dimensional block, not subjected to constrained flow conditions, using AIRFLO3D data

Figure 4.6 Plot for Drag Coefficient variation with l/h ratio using AIRFLO3D data for Case (A)
Figure 4.7 Plot for Drag Correction using PHOENICS data for Case (A)
Figure 4.8 Plot for Drag Correction using AIRFLO3D data for Case (A)
The blockage correction equations for this case are,

\[ C_{D_{PRBOSN}} = 0.0838C_{D_{appr}} + 1.0318. \]
\[ C_{D_{abortion}} = 0.2419C_{D_{appr}} + 1.7534. \] (4.2)
4.2 Case (B) Two Dimensional Flow around a rectangular section subjected to a constrained flow field in a wind tunnel test section

The computational domain in this case also, is vertically symmetric. A Constant velocity profile is applied at the inlet. This produces a vertically symmetric flow structure around the blockage. For this reason, the pressure signature and the velocity profile are plotted for the data acquired along a line drawn longitudinally in the flow direction, vertically positioned on top of the rectangular blockage, i.e., along line AB in Figure 4.1. Results have been obtained from both PHOENICS and AIRFLO3D.

From the plot for pressure signature using PHOENICS data, one can see a gradual increase in the positive pressure from the test section entrance, up to approximately one-third blockage length, ahead of the blockage. Thereafter, the pressure takes a rapid increase till the starting edge of the blockage. Beyond this point, it steeply falls down below zero pressure mark, through a short distance on the blockage. Then an increase in the pressure is seen, which stabilizes along the surface of the blockage and a little beyond the blockage. Past that point, the pressure further increases and then stabilizes at zero pressure way up to the exit of the test section. An insignificant variation in pressure is seen for varying blockage ratios; but along the surface of the blockage and a little beyond it, the magnitude of pressure decreases with increasing blockage ratio. The same explanation does not hold good for the data obtained from AIRFLO3D.

According to AIRFLO3D, the pressure at the test section entrance is approximately at zero mark. This pressure as detected in PHOENICS increases up to a distance, approximately one-third the blockage length ahead of the blockage. Then the
pressure steeply drops beyond this up to a point a little beyond the blockage starting point. Then the pressure again increases along the blockage surface, but does not remain stable. Beyond the end point of the blockage, the pressure drops again in the wake region and suddenly increases beyond the wake region. The pressure beyond this gradually decreases and stabilizes at the test section exit. A considerable variation can be seen in the pressure signature can be seen with the change in the blockage ratio. All along the longitudinal distance considered, at any point, the pressure signature magnitude decreases with increase in the blockage ratio. The variations in pressure signature can be clearly seen in Figures 4.9 and 4.11.

From the plot in Figure 4.10, the velocity profile from PHOENICS data, it can be seen that the velocity magnitude decreases gradually up to a point in front of the blockage. A little distance beyond the starting point of the blockage, a minute increase in the velocity is seen; but beyond this point, it decreases again along the blockage length, with a little increase, in the wake region beyond the blockage. Past the wake, the velocity gradually increases and stabilizes finally at the exit. There is not much variation seen in the velocity with varying blockage ratio. On the other hand, in case of the velocity profile obtained from AIRFLO3D, Figure 4.12, there is a constant decrease in the velocity till the starting point of the blockage. The velocity then, suddenly increases and the trend continues along the blockage and into the wake region, where the velocity is highest in magnitude. Beyond the wake region, the velocity drops suddenly and with a gradual increase, it stabilizes as the exit of the test section approaches.
Figure 4.9 Plot for Pressure Signature using PHOENICS data for Case (B)
Figure 4.10 Plot for Velocity Profile using PHOENICS data for Case (B)
Plot for the Pressure Signature at the top/bottom line of a Two Dimensional Rectangular Block using the data obtained from AIRFLO3D

Figure 4.11 Plot for Pressure Signature using AIRFLO3D data for Case (B)
Figure 4.12 Plot for Velocity Profile using AIRFLO3D data for Case (B)
The increase in pressure in front of the blockage, is caused due to the stagnation of flow at the front of the blockage. The formation of stagnation point causes an increase in the pressure in its vicinity, gradually decreasing to attain normal value, away from the point. The steep decrease in pressure, immediately after that, is caused due to the sudden change in direction of flow and the increase in velocity at the front edge of the blockage. Beyond this point, the pressure is not much effected by the change in flow, due to which, a gradual increase and subsequent stabilization of pressure is seen, as predicted by PHOENICS. But beyond the blockage, in the vicinity of the wake, AIRFLO3D has predicted a sudden dip in pressure, which shows that the pressure is effected by the sudden decrease in velocity in the wake region. This might be attributed the Kato and Launder K-ε model, used in AIRFLO3D code.

The velocity vector plot for this case (See Appendix), shows that the flow separation occurs at the top of the block and reattaches at a point around the midway of the blockage. PHOENICS does not produce a wake bubble at the top of the blockage. This might be because the K-ε model with wall conditions was not able to predict it. The Kato and Launder model used in AIRFLO3D has, predicted the wake as indicated in (27). It can also be seen that the K-ε model used in PHOENICS, shows the effect of the wake even at the test section exit, which is not the same with AIRFLO3D. The wake is confined to a very small region behind the blockage and the flow past the wake tends to get normal.

The drag coefficients are plotted for different blockage ratios. Drag coefficients obtained from both PHOENICS and AIRFLO3D are plotted for variation with Blockage
ratio and are compared. The Blockage correction for the AIRFLO3D is obtained by correcting the drag coefficient with respect to those of PHOENICS. Figures 4.13 and 4.14 show the drag variations with respect to blockage ratios for PHOENICS and AIRFLO3D, respectively. Figure 4.15 shows the blockage correction for the AIRFLO3D drag.

From the plots, it can be seen that the drag coefficient obtained from both softwares, increase with increasing blockage ratio, except that AIRFLO3D has higher drag coefficients compared to those of PHOENICS. The variation in the drag coefficients with blockage ratio is considerably higher in case of AIRFLO3D.

The Blockage correction equation obtained is

\[ C_{D_{\text{AIRFLO3D}}} = 2.2935 \, C_{D_{\text{PHOENICS}}} - 0.2216. \] (4.3)
Figure 4.13 Plot for Drag Coefficient variation with blockage ratio using PHOENICS data
Figure 4.14 Plot for Drag Coefficient variation with blockage ratio using AIRFLO3D data
Figure 4.15 Plot for Drag Correction using AIRFLO3D and PHOENICS data for Case(B)
4.3 Case (C) Three-dimensional flow around a rectangular block placed on the floor in a wind tunnel test section

The three-dimensional rectangular block is placed longitudinally exactly in the midway of the width of the test section. The layout of this case is shown in figure (3.3). This case is dealt differently in PHOENICS and AIRFLO3D. The plots for pressure signature and the velocity profile are drawn with the data obtained along a line drawn in the plane of longitudinal symmetry, passing over the top surface of the blockage and normal to the cross section of the wind tunnel test section as well as the blockage as shown along the line AB in Figure 4.16. In PHOENICS, the whole test section domain is considered otherwise compared to AIRFLO3D where only half the test section is taken into account.

![Figure 4.16 Line along which the Pressure and Velocity Distributions are obtained in case of Three Dimensional rectangular block](image)

The pressure signature plot from PHOENICS data, Figure 4.17, indicates a gradual increase in the value from the test section entrance to a considerable distance ahead of the blockage. Then the pressure drops down sharply to a least value beyond this
point until a point just after the front wall of the blockage is reached. From this, the pressure again rises to an intermediate level and partially stabilizes over the blockage surface. The pressure is found to decrease again a little at the end of the blockage, but increases in the wake region. The pressure past the wake is found to gradually tending to stabilize. For the three blockage ratios, beyond the front wall of the blockage, the pressure is found to decrease with increasing blockage ratio, except at the exit, that the three cases end up at zero pressure.

As in the case of AIRFLO3D, refer Figure 4.19, the pattern of the pressure signature matches with that of the PHOENICS and that the pressure decreases with increasing blockage ratio. But the difference between the two is that, beyond the wake region the difference in pressure magnitude is seen to exist till the test section exit point.

If one looks at the velocity profile for both PHOENICS and AIRFLO3D, they match very well in most of the regions. The velocity is found to gradually decrease in magnitude through the entrance to the vicinity of the stagnation point. But at the front edge region, a hump in the velocity profile is seen followed by a considerable reduction in magnitude which almost dies down to zero in the midway of the blockage length. The velocity then rises gradually beyond that. With the little change in course of the profile at the wake region, the velocity tends to attain the initial velocity conditions at exit.

The velocity profile is found to have not effected by the change in the blockage ratio in case of AIRFLO3D. In case of PHOENICS, the velocity is found to decrease with increasing blockage ratio, in the regions just beyond the front face of the blockage and beyond the rear face of the blockage. Figures 4.18 and 4.20 depict the velocity profiles.
The pressure drop in the region beyond the front face of the blockage is due to the increase in velocity at the front edge. The velocity increase is caused due to the fact that the flow enters a contracted region over the blockage. The flow velocity increase, extend over the blockage at a considerable height.
Plot for Pressure signature at the top surface along the symmetric line for a three dimensional Rectangular Block using the data obtained from PHOENICS

Figure 4.17 Plot for Pressure Signature using PHOENICS data for Case (C)
Plot for Velocity Profile at the top surface along the symmetric line for a Three Dimensional Rectangular Block using the data obtained from PHOENICS

Figure 4.18 Plot for Velocity Profile using PHOENICS data for Case (C)
Plot for Pressure Signature at the top surface along the symmetric line for a three
Dimensional Rectangular Block using the data obtained from AIRFLO3D

Figure 4.19 Plot for Pressure Signature using AIRFLO3D data for Case (C)
Plot for Velocity Profile at the top surface along the symmetric line for a three Dimensional Rectangular Block using the data obtained from AIRFLO3D

Figure 4.20 Plot for Velocity Profile using AIRFLO3D data for Case (C)
The reduction in the flow velocity just over the blockage may be due to the formation of boundary layer at the top surface. In PHOENICS, the separation of this boundary layer occurs farther on the blockage surface (rear end of the blockage) compared to the case in AIRFL03D. As a result, an early rise in flow velocity on the blockage is observed in AIRFL03D. The Kato and Launder model, which avoids over-prediction of the kinetic energy, which in turn is the energy due to the velocity of the fluid particles, should be the reason for the early rise in the velocity from minimum. The low Reynolds number K-ε model used in PHOENICS does not over-predict the results as in the ordinary K-ε model. But it provides a more realistic flow structure.

The pressure signature is observed to take a small dip again at the rear end of the blockage. This can be attributed to the flow separation that occurs at the region in case of PHOENICS. The same type of dip is observed in AIRFL03D much earlier, to which the explanation is as above.

The faint dip in the velocity magnitude at the rear edge of the blockage is because; the region beyond this is the vicinity of the wake bubble. Beyond the wake bubble, the velocity tends to pick, which is caused due to the effect of previous velocity increase at the front edge. This can be seen in the velocity vector plot provided in the Appendix.

The drag coefficients are plotted for different blockage ratios. Drag coefficients obtained from both PHOENICS and AIRFL03D are plotted for variation with Blockage ratio and are compared. The Blockage correction for the AIRFL03D is obtained by correcting the drag coefficient with respect to those of PHOENICS. Figures 4.21 and 4.22
show the drag variations with respect to blockage ratios for PHOENICS and AIRFLO3D, respectively. From the plots, it can be seen that the drag coefficient obtained from both softwares, increase with increasing blockage ratio, except that AIRFLO3D has higher drag coefficients compared to those of PHOENICS. The variation in the drag coefficients with blockage ratio is considerably higher in case of AIRFLO3D.

The Blockage correction equations are taken as quadratic fit for the drag coefficients plotted against blockage ratio. The equations are in the form

\begin{align*}
C_{D_{\text{AIRFLO3D}}} &= 0.0012 \left( \frac{A_m}{A_w} \right)^2 + 0.0269 \left( \frac{A_m}{A_w} \right) + 1.511. \\
C_{D_{\text{PHOENICS}}} &= 0.0016 \left( \frac{A_m}{A_w} \right)^2 - 0.0241 \left( \frac{A_m}{A_w} \right) + 1.098.
\end{align*}

(4.4)  
(4.5)

From the above equations, it can be seen that the drag variation with blockage ratio is almost linear.
Figure 4.21 Plot for Drag Coefficient varying with Blockage Ratio using PHOENICS data

\[ y = 0.0016x^2 - 0.0241x + 1.096 \]
Figure 4.22 Plot for Drag Coefficient varying with Blockage Ratio using AIRFLO3D data
4.4 Case (D) Three Dimensional flow around a top-front edge tapered rectangular blockage placed close to the floor of a wind tunnel test section

In this case, the blockage is located at a certain height above the wind tunnel test section floor. The flow is asymmetric vertically which is not the same situation as in cases (A) and (B). So the pressure signatures and velocity profiles along the upper as well as the lower surfaces of the blockage are needed to be analyzed. The lines along which the plots are drawn are placed along the plane of longitudinal symmetry and normal to the cross sectional area of the test section.

![Diagram](image)

Figure 4.23 Lines along which the Pressure and Velocity Distributions are obtained in case of Three Dimensional top-edge tapered block

The pressure signature along the top of the blockage is analyzed first i.e., line AB in Figure 4.23. In this case, the trend in the pressure signature profiles obtained from both PHOENICS and AIRFLO3D is same. But in front of the blockage front end, the pressure rise in PHOENICS is observed to be more than that in AIRFLO3D. Even the drop in pressure beyond that point above the slant surface is more compared to that in

85
AIRFLO3D. The signature above the flat surface of the blockage is observed to be stabilizing a little. But beyond the rear end of the blockage, there is a dip in the pressure and a gradual rise, attaining a stabilized state at the test section exit. The pressure at the test section exit is found to be zero, irrespective of the blockage ratio, in the case of PHOENICS. But there is a considerable difference in the pressure signature with variation in blockage ratio in case of AIRFLO3D. The pressure is observed to decrease with increase in the blockage ratio. From the test section entrance to the blockage front end, the pressure variation with blockage is not observed to as significant extent. Figures 4.24 and 4.26 depict the variations in pressure signature.

The velocity profiles for PHOENICS and AIRFLO3D differ to a certain extent. According to the PHOENICS data, Figure 4.25, a velocity drop is seen in the vicinity of the stagnation point at the blockage front end. Beyond that, an increase is seen in velocity up to the intersection of the slanted surface and the top horizontal surface of the blockage. Then again, velocity decreases gradually till the blockage rear end. Then a sudden decrease in the velocity in the wake region, followed by stabilization of flow at the test section exit, is observed.

The flow model is more detailed in AIRFLO3D, Figure 4.27. The velocity drops down up to the front end of the blockage. Past this point, due to an increase in the flow velocity at the front edge, there is a corresponding increase in the flow above this point. But again, due to the formation of the boundary layer on the slant surface, velocity drop in the region occurs. At the point of intersection of the slant surface and the horizontal plane at the top, velocity increases, due to the change in flow direction at the point and
flow contraction beyond it. However the flow contraction is observed to be gradual, which does not effect much, the increasing trend of the velocity, till the rear end of the blockage is reached. But at the vicinity of the wake bubble, the velocity drops down and before the exit is reached, the flow tends to stabilize.
Plot for Pressure Signature at the top surface along the symmetric line for a top-front edge tapered Block using the data obtained from PHOENICS

Figure 4.24 Plot for Pressure Signature at the top of the blockage, using PHOENICS data for Case (D)
Figure 4.25 Plot for Velocity Profile at the top of the blockage, using PHOENICS data for Case (D)
Plot for Pressure Signature at the top surface along the symmetric line for a top-front edge tapered Block using the data obtained from AIRFLO3D

Figure 4.26 Plot for Pressure Signature at the top of the blockage, using AIRFLO3D data for Case (D)
Figure 4.27 Plot for Velocity Profile at the top of the blockage, using AIRFLO3D data for Case (D)
At the bottom surface of the blockage, the pattern of the pressure signature follows in the same lines as that of the case (C), i.e., along the line CD in Figure 4.23. The pressure drops down steeply after a faint increase ahead of the front end of the blockage. According to PHOENICS, Figure 4.28, the pressure it recovers along the bottom surface, drops down a little in the wake region and increases again beyond the wake region. At the exit, the pressure attains zero value, with the gradient of the pressure curve tending to zero. On the other hand, AIRFLO3D, as usual, predicts the variation of pressure gradient to be significant, with the change in blockage ratio, thought the trend followed by the pressure line is same as that of PHOENICS, Figure 4.30.

The velocity at the bottom also increases till the front end of the blockage and beyond that it decreases due to the formation of a boundary layer on the bottom surface. Then at the separation point, it increase. But suddenly decreases, as it reaches the wake bubble region. The flow again increases beyond the wake region up to the test section exit. The flow profiles for PHOENICS and AIRFLO3D are depicted in Figures 4.29 and 4.31.
Figure 4.28 Plot for Pressure Signature at the bottom of the blockage, using PHOENICS data for Case (D)
Figure 4.29 Plot for Velocity Profile at the bottom of the blockage, using PHOENICS data for Case (D)
Figure 4.30 Plot for Pressure Signature at the bottom of the blockage, using AIRFLO3D data for Case (D)
Plot for Velocity Profile at the bottom surface along the symmetric line for a top-front edge tapered Block using the data obtained from AIRFLO3D

Figure 4.31 Plot for Velocity Profile at the bottom of the blockage, using AIRFLO3D data for Case (D)
The drag coefficients are plotted for different blockage ratios. Drag coefficients obtained from both PHOENICS and AIRFLO3D are plotted for variation with Blockage ratio and are compared. The Blockage correction for AIRFLO3D is obtained by correcting the drag coefficient with respect to those of PHOENICS. The Figures 4.32 and 4.33 show the drag variations with respect to blockage ratios for PHOENICS and AIRFLO3D, respectively. From the plots, it can be seen that the drag coefficient obtained from both softwares, increase with increasing blockage ratio, except that AIRFLO3D has higher drag coefficients compared to those of PHOENICS. The variation in the drag coefficients with blockage ratio is considerably higher in case of AIRFLO3D.

The Blockage correction equations are taken as quadratic fit for the drag coefficients plotted against blockage ratio. The equations are in the form

\[
C_D^{\text{AIRFLO3D}} = 0.0048\left(\frac{A_m}{A_w}\right)^2 - 0.0214\left(\frac{A_m}{A_w}\right) + 1.091 \quad (4.6)
\]

\[
C_D^{\text{PHOENICS}} = 0.0003\left(\frac{A_m}{A_w}\right)^2 + 0.0157\left(\frac{A_m}{A_w}\right) + 0.5698. \quad (4.7)
\]

From the above equations, it can be seen that the drag variation with blockage ratio is almost linear.
Figure 4.32 Plot for Drag Coefficient variation with Blockage Ratio using PHOENICS data
Plot for Drag variation with Blockage Ratio for a top-front edge tapered block using AIRFLO3D data

\[ y = 0.0048x^2 - 0.0214x + 1.091 \]

Figure 4.33 Plot for Drag Coefficient variation with Blockage Ratio using AIRFLO3D data
### 4.5 Grid Refinement Study on Case (C) for 10% Blockage Ratio

The grid size of the mesh has been varied and the pressure signature and velocity plots have been obtained and a comparison study has been done. The size of the grid was doubled to that of the actual grid size considered. The grid sizes of the coarse mesh and the fine mesh are presented in Table 4.1.

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>Coarse</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Size</td>
<td>109200</td>
<td>219920</td>
</tr>
<tr>
<td>Mesh Size</td>
<td>65470</td>
<td>133768</td>
</tr>
</tbody>
</table>

The mesh size when seen in PHOENICS is found to have no significant effect on the pressure signature and the velocity profiles, except that, for the finer mesh the pressure magnitude near the front end of the blockage is more steep compared to that of the coarser mesh. The pressure signature and the velocity profile plots obtained from PHOENICS are shown in Figures 4.34 and 4.35. On the other hand, AIRFLO3D predicts a significant change in the pressure signature as well as the velocity profile plots, though there is no effect of mesh refinement on the flow in the front portion of the test section in front of the blockage. The plots for these parameters are shown in Figures 4.36 and 4.37.
Pressure Signature variation with Grid Refinement for Case (C) for 10% Blockage Ratio

Figure 4.34 Plot for variation of Pressure Signature with Mesh Refinement using PHOENICS Data
Figure 4.35 Plot for variation of Velocity Profile with Mesh Refinement using PHOENICS Data
Figure 4.36 Plot for variation of Pressure Signature with Mesh Refinement using AIRFLO3D Data
Figure 4.37 Plot for variation of Velocity Profile with Mesh Refinement using AIRFLO3D Data
CFD flow simulations have successfully been performed using PHOENICS and AIRFL03D for different blockages placed in wind tunnel test sections. The effects of wind tunnel blockages on the pressure and velocity distributions along the flow direction have been predicted. The blockage effects due to varying test section height have been found to affect the flow parameters, significantly. Blockage corrections from the drag correction equations have been proposed for the data obtained from PHOENICS and AIRFL03D for both two dimensional and three dimensional cases. The drag coefficient variation with blockage $b/h$ ratio, in case of a two dimensional unconstrained flow, reasonably agree with the experimental results. The AIRFL03D drag predictions are seen to have yielded higher values compared to PHOENICS and the experimental drag coefficients. AIRFL03D is also found to show significant variations in the drag coefficients compared to PHOENICS. The blockage correction equations obtained, provide a good correlation for drag coefficient and the test section height. The velocity vector plots provide as good source for differentiating the turbulence models used in PHOENICS and AIRFL03D. The grid independent study performed predicted a negligible influence of mesh size on the results obtained by PHOENICS but the results obtained from AIRFL03D have a significant effect due to mesh refinement.

Further study is necessary for determining the reason for higher drag predictions in AIRFL03D. The computational experiments conducted in this work can be extended
to a wide range of blockage geometries. As a part of future work, similar type of experiments if conducted on the blockage geometries used by MIRA, would provide a strong basis for future development in the study of blockage effects involved in wind tunnel testing on automobiles. The study can also be extended to determine the dependence of blockage effects on mesh shape, unsteady conditions and different velocity profiles.
REFERENCES


3. “Closed-Test-Section Wind Tunnel Blockage Corrections for Road Vehicles.” SAE International. SP-1176


APPENDIX A

VELOCITY VECTOR PLOTS OBTAINED FROM PHOENICS

Figure A.1 Two dimensional rectangular block; 5% Blockage ratio
Figure A.2 Two dimensional rectangular block; 10% Blockage ratio

Figure A.3 Two dimensional rectangular block; 15% Blockage ratio
Figure A.4 Three dimensional rectangular block; 5% Blockage ratio
Figure A.5 Three dimensional rectangular block; 10% Blockage ratio

Figure A.6 Three dimensional rectangular block; 15% Blockage ratio
<table>
<thead>
<tr>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.396E+00</td>
</tr>
<tr>
<td>1.309E+00</td>
</tr>
<tr>
<td>1.222E+00</td>
</tr>
<tr>
<td>1.135E+00</td>
</tr>
<tr>
<td>1.049E+00</td>
</tr>
<tr>
<td>9.617E-01</td>
</tr>
<tr>
<td>8.750E-01</td>
</tr>
<tr>
<td>7.882E-01</td>
</tr>
<tr>
<td>7.014E-01</td>
</tr>
<tr>
<td>6.146E-01</td>
</tr>
<tr>
<td>5.278E-01</td>
</tr>
<tr>
<td>4.410E-01</td>
</tr>
<tr>
<td>3.542E-01</td>
</tr>
<tr>
<td>2.674E-01</td>
</tr>
<tr>
<td>1.806E-01</td>
</tr>
<tr>
<td>9.384E-02</td>
</tr>
<tr>
<td>7.050E-03</td>
</tr>
</tbody>
</table>

Figure A.7 Three dimensional top-front edge tapered block; 5% Blockage ratio
Figure A.8 Three dimensional top-front edge tapered block; 5% Blockage ratio

Figure A.9 Three dimensional top-front edge tapered block; 15% Blockage ratio
Figure B.1 Two dimensional rectangular block; 5% Blockage ratio
Figure B.2 Two dimensional rectangular block; 10% Blockage ratio

Figure B.3 Two dimensional rectangular block; 15% Blockage ratio
Figure B.4 Three dimensional rectangular block; 5% Blockage ratio
Figure B.5 Three dimensional rectangular block; 10% Blockage ratio

Figure B.6 Three dimensional rectangular block; 15% Blockage ratio
Figure B.7 Three dimensional top-front edge tapered block; 5% Blockage ratio
Figure B.8 Three dimensional top-front edge tapered block; 10% Blockage ratio

Figure B.9 Three dimensional top-front edge tapered block; 15% Blockage ratio
PERMISSION TO COPY

In presenting this thesis in partial fulfillment of the requirements for a master's degree at Texas Tech University or Texas Tech University Health Sciences Center, I agree that the Library and my major department shall make it freely available for research purposes. Permission to copy this thesis for scholarly purposes may be granted by the Director of the Library or my major professor. It is understood that any copying or publication of this thesis for financial gain shall not be allowed without my further written permission and that any user may be liable for copyright infringement.

**Agree** (Permission is granted.)

____________________________________  ____________________________  
Student Signature  Date

**Disagree** (Permission is not granted.)

____________________________________  ____________________________  
Student Signature  Date