

DETACHED EDDY SIMULATIONS OF A SIMPLIFIED TRACTOR-TRAILER GEOMETRY

Except where reference is made to the work of others, the work described in this thesis is my own or was done in collaboration with my advisory committee. This thesis does not include proprietary or classified information.

Harshavardhan Ghuge

Certificate of Approval:

Brian Thurow
Assistant Professor
Aerospace Engineering

Christopher Roy, Chair
Assistant Professor
Aerospace Engineering

Anwar Ahmed
Associate Professor
Aerospace Engineering

Joe F. Pittman
Interim Dean
Graduate School

DETACHED EDDY SIMULATIONS OF A SIMPLIFIED TRACTOR-TRAILER GEOMETRY

Harshavardhan Ghuge

A Thesis

Submitted to

the Graduate Faculty of

Auburn University

in Partial Fulfillment of the

Requirements for the

Degree of

Master of Science

Auburn, Alabama

May 10, 2007

DETACHED EDDY SIMULATIONS OF A SIMPLIFIED TRACTOR-TRAILER GEOMETRY

Harshavardhan Ghuge

Permission is granted to Auburn University to make copies of this thesis at its discretion, upon the request of individuals or institutions and at their expense. The author reserves all publication rights.

Signature of Author

Date of Graduation

VITA

Harshavardhan Ghuge, son of Arvind and Vaishali Ghuge was born on July 02, 1981, in Pune, India. He attended Maharashtra Institute of Technology, Pune and graduated in June 2004 with the degree of Bachelor of Mechanical Engineering. He joined the Masters program in the department of Aerospace Engineering at Auburn University in August 2004.

THESIS ABSTRACT

DETACHED EDDY SIMULATIONS OF A SIMPLIFIED TRACTOR-TRAILER GEOMETRY

Harshavardhan Ghuge

Master of Science, May 10, 2007

(B.E., Maharashtra Institute of Technology, Pune University, 2004)

116 Typed Pages

Directed by Christopher J. Roy

This thesis presents a comparison of the results from the Detached Eddy Simulation (DES) turbulence model used for modeling flow over a $1/8^{th}$ scale simplified model of a tractor-trailer geometry (or the Ground Transportation System) with the experimental data. The main aim of this study is to assess the DES model for validation purpose. From the results it is concluded that the DES model gives very good agreement with the experimental data for the global quantities like drag but does not predict the details of the wake structure correctly. The reason for the inaccuracy in the prediction may be originating from the lack of knowledge about the transfer of information between the Reynolds-Averaged Navier-Stokes region (where the turbulence is modeled) and the Large Eddy Simulation region (where the turbulence is simulated).

ACKNOWLEDGMENTS

I would like to acknowledge the U.S. Department of Energy FreedomCAR and Vehicle Technologies (CAR stands for Cooperative Automotive Research) for funding this work at Auburn University. I would like to thank my adviser Dr. Christopher J. Roy, Assistant Professor, Aerospace Engineering Department for his guidance and support throughout the research and toward the completion of this thesis.

Many thanks to Dr Matthew F. Barone of the Aerosciences and Compressible Fluid Mechanics Department from Sandia Laboratories for his insightful guidance during this research. I would also like to thank Dr. Brian Thurow, Assistant Professor, Aerospace Engineering Department, for his assistance during my research and for serving as a committee member. I am thankful to Dr. Anwar Ahmed, Associate Professor, Aerospace Engineering Department for serving as a committee member.

Most importantly, I am grateful to my mother for patiently supporting me throughout my education.

Style manual or journal used LATEX: A Document Preparation System by Leslie Lamport (together with the style known as “aums”)

Computer software used TEX (specifically LATEX), MATLAB 7.0.4, Tecplot and the departmental style-file aums.sty

TABLE OF CONTENTS

LIST OF FIGURES	x
LIST OF TABLES	xiv
NOMENCLATURE	xv
1 INTRODUCTION	1
1.1 Research Objective	3
1.2 Introduction to Turbulence Models	4
1.2.1 Reynolds Averaged Navier-Stokes Equations	4
1.2.2 Introduction to Large Eddy Simulation (LES)	7
1.2.3 Introduction to Hybrid RANS/LES Models	8
1.3 Literature Review	8
1.3.1 Experimental Research	9
1.3.2 Computational Research	13
2 PROBLEM DESCRIPTION	24
2.1 Wind Tunnel Experiment	24
2.1.1 Surface Pressures	26
2.1.2 Drag, Lift and Side Force Measurements	27
2.1.3 Oil-film Interferometry (OFI), Hot-film Anemometry and Particle Image Velocimetry (PIV)	27
3 SIMULATION APPROACH	28
3.1 Governing Equations	28
3.2 The DES Model	28
3.3 The SACCARA Code	32
3.4 Discretization of the Convective Terms	33
3.4.1 Finite Volume Method	33
3.4.2 Upwind Schemes	33
3.5 Discretization of the Diffusion Terms	35
3.6 Temporal Discretization	35
3.7 Grid Generation Approach	35
3.8 Domain Decomposition and Load Balancing	36
3.9 Boundary Conditions used for the Simulations	37
3.10 Characteristic Scales	38

4	NUMERICAL ACCURACY	43
4.1	Statistical Convergence	43
4.2	Initial Transients	51
4.3	Effect of Number of Sub-iterations used for Iterative Convergence	57
4.4	Effect of Time Step ($\Delta\tau_s$)	60
4.5	Effect of the Mesh Size on DES	63
5	RESULTS AND DISCUSSION	67
5.1	Contour and Streamline Plots	67
5.2	Velocity Profiles	71
5.3	Pressure Distribution Profile	76
5.4	Drag, Lift and Side Force Coefficients	79
5.5	Vortex Core Locations	80
5.6	Turbulent Eddies in the Wake	82
5.7	Power Spectra of the Unsteady Pressure Signal	83
6	CONCLUSION	88
7	RECOMMENDATIONS	91
	BIBLIOGRAPHY	92
	APPENDICES	97
A	FORTRAN CODE FOR CREATING ANIMATIONS USING TECPLOT	98

LIST OF FIGURES

2.1	The GTS model	25
3.1	Reference static pressure comparison between the coarse mesh simulations and the experiment	40
3.2	Coarse mesh structure near the truck base	41
3.3	Medium mesh structure near the truck base	42
4.1	Time windows used for calculating statistically converged results	45
4.2	Location of pressure and velocity profiles used for statistical convergence .	46
4.3	Pressure distribution along line T and corresponding to different time windows	47
4.4	Velocity profiles along line A and corresponding to different time windows	48
4.5	Error in the pressure distribution obtained from smaller time windows with respect to the pressure distribution obtained from the biggest time window	49
4.6	Error in the u/V_{ref} velocity profiles obtained from smaller time windows with respect to the u/V_{ref} velocity profile obtained from biggest time window	49
4.7	Error in the v/V_{ref} velocity profiles obtained from smaller time windows with respect to the v/V_{ref} velocity profile obtained from biggest time window	49
4.8	Error in the u_{rms}/V_{ref} velocity profiles obtained from smaller time win- dows with respect to the u_{rms}/V_{ref} velocity profile obtained from biggest time window	49
4.9	Time windows used for estimating period of initial transients	52
4.10	Pressure distribution along line T and corresponding to different time windows	53

4.11	Velocity profiles along line A and corresponding to different time windows	54
4.12	Error in the pressure distribution from all time windows with respect to the pressure distribution obtained from the window farthest along time-axis	55
4.13	Error in the u/V_{ref} profiles from all time windows with respect to the u/V_{ref} profile obtained from the window farthest along time-axis	55
4.14	Error in the v/V_{ref} profiles from all time windows with respect to the v/V_{ref} profile obtained from the window farthest along time-axis	56
4.15	Error in the u_{rms}/V_{ref} profiles from all time windows with respect to the u_{rms}/V_{ref} profile obtained from the window farthest along time-axis . . .	56
4.16	Pressure distribution along line T and corresponding to different number of sub-iterations	58
4.17	Velocity profiles along line A and corresponding to different number of sub-iterations	59
4.18	Pressure distribution along line T and corresponding to different time-steps	61
4.19	Velocity profiles along line A and corresponding to different time-steps . .	62
4.20	Medium mesh vertical streamwise contour plot	64
4.21	Coarse mesh vertical streamwise contour plot	64
4.22	Pressure distribution along line T obtained from the coarse and medium mesh simulations	65
4.23	Velocity profiles along line A obtained from the coarse and medium mesh simulations	66
5.1	Planes used for comparison of velocity data	67
5.2	Vertical streamwise PIV window in the experiment	68
5.3	Vertical streamwise contour plot from the medium mesh simulations . . .	68
5.4	Horizontal streamwise PIV window in the experiment ($y/W = 0.696$) . .	69
5.5	Horizontal streamwise contour plot from the medium mesh simulations ($y/W = 0.709$)	69

5.6	Horizontal streamwise PIV window in the experiment ($y/W = 1.044$) . .	70
5.7	Horizontal streamwise contour plot from the medium mesh simulations ($y/W = 1.05$)	70
5.8	Comparison of u/V_{ref} profiles from the experiment and simulations at $y/W = 0.348$ and $z/W = 0$	72
5.9	Comparison of u/V_{ref} profiles from the experiment and simulations at $y/W = 0.696$ and $z/W = 0$	72
5.10	Comparison of u/V_{ref} profiles from the experiment and simulations at $y/W = 1.044$ and $z/W = 0$	73
5.11	Comparison of v/V_{ref} profiles from the experiment and simulations at $y/W = 0.348$ and $z/W = 0$	73
5.12	Comparison of v/V_{ref} profiles from the experiment and simulations at $y/W = 0.696$ and $z/W = 0$	73
5.13	Comparison of v/V_{ref} profiles from the experiment and simulations at $y/W = 1.044$ and $z/W = 0$	74
5.14	Comparison of w/V_{ref} profiles from the experiment and simulations at $y/W = 0.348$ and $z/W = 0$	75
5.15	Comparison of w/V_{ref} profiles from the experiment and simulations at $y/W = 0.696$ and $z/W = 0$	75
5.16	Comparison of w/V_{ref} profiles from the experiment and simulations at $y/W = 1.044$ and $z/W = 0$	75
5.17	Comparison of pressure distribution along base of the truck from the ex- periment and the medium mesh simulations	77
5.18	Comparison of pressure distribution on the front surface of the GTS model	78
5.19	Comparison of pressure distribution on the top surface of the GTS model	78
5.20	Comparison of pressure distribution along the right side of the GTS model	78
5.21	Comparison of pressure distribution on the bottom surface of the GTS model	78

5.22	Coarse mesh mean and instantaneous drag coefficient	80
5.23	Medium mesh mean and instantaneous drag coefficient	80
5.24	Location of vortex cores in vertical streamwise plane ($z/W = 0.0$)	81
5.25	Location of vortex cores in horizontal streamwise plane ($y/W = 0.696$)	81
5.26	Location of vortex cores in horizontal streamwise plane ($y/W = 1.044$)	81
5.27	Vortical structures resolved using LES in the wake of the GTS on the medium mesh	82
5.28	Fast Fourier transform of the pressure signal from experimental data (x/W $= 7.647$, $y/W = 0.63$ and $z/W = -0.46$)	85
5.29	Fast Fourier transform of the pressure signal from coarse mesh simulations ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$)	86
5.30	Fast Fourier transform of the pressure signal from medium mesh simula- tions ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$)	86
5.31	Fast Fourier transform of the pressure signal in the horizontal streamwise plane from the coarse mesh simulations	86
5.32	Fast Fourier transform of the pressure signal in the horizontal streamwise plane from the coarse mesh simulations	86
5.33	Fast Fourier transform of the pressure signal from the vertical streamwise plane in the coarse mesh and near the bottom surface of the GTS model	87
5.34	Fast Fourier transform of the pressure signal from the vertical streamwise plane in the coarse mesh and near the top surface of the GTS model	87

LIST OF TABLES

1.1	Effect of drag-reduction concepts on wind-averaged drag [5].	11
2.1	Wind tunnel conditions [4].	25
2.2	Experimental values of pressure coefficient at different z/W locations along the y/W axis of the trailer base [4].	26
3.1	Coarse and medium mesh	37
3.2	Reference pressure comparison	38
3.3	Characteristics scales	39
4.1	Absolute error with respect to biggest time window	50
4.2	Absolute error with respect to time window farthest along the time-axis .	56
5.1	Total uncertainty values of the velocity and pressure from the simulations	74
5.2	Comparison of the drag, side and lift force coefficients	79

NOMENCLATURE

English Symbols

b	distance between square cylinder bottom face and the channel floor (m)
c	c^{th} processor
$c_{b1}, c_{b2}, c_{v1}, c_{w1}$	
c_{w2}, c_{w3}	closure coefficients in Spalart-Allmaras model
C_D	coefficient of drag
C_{DES}	detached eddy simulation constant
C_{Dmean}	mean value of coefficient of drag
C_k	coefficient of thermal conductivity (W/m/K)
C_L	coefficient of lift
C_{Lmean}	mean value of coefficient of lift
C_p	time-averaged coefficient of pressure
c_p	specific heat at constant pressure (J/kg/K)
$(C_p)_{error}$	error in the C_p profile
C_S	coefficient of side force

CHAPTER 1

INTRODUCTION

Class 8 tractor-trailers carry the major load of transportation in the trucking industry and are used for transportation of most of the commodities across the United States of America. The weight of a modern Class 8 tractor-trailer can be up to 80,000 pounds. At a common highway speed of 70 mph 65% of the total engine output is consumed in overcoming the aerodynamic drag. Also the energy used in overcoming the aerodynamic drag increases as the vehicle speed increases [1]. During late 1970's and early 1980's improvement in truck aerodynamics to reduce fuel consumption led to the development of the following technologies [2]:

1. Cab shaping
2. Cab-mounted deflectors
3. Trailer front-end fairings
4. Cab side extenders
5. Body front-edge rounding
6. Tractor-trailer gap seals
7. Trailer side skirts
8. Rear-boat-tailing

The first five items are called the first generation drag reduction devices and are already in use. The last three items are called second generation drag reducing devices and are not used widely. A typical Class 8 tractor-trailer has a wind-averaged drag coefficient of $C_D = 0.6$. A wind-averaged drag coefficient is a drag coefficient that takes into account the effects of wind blowing from all the directions as is the case when a vehicle is moving on the road. The present day truck drag coefficients can be reduced by as much as 50% [1].

According to the statistics presented in [3] there were 2.2 million trucks on the US roads in the year 2003. Each truck traveled approximately 62,900 miles with an average fuel efficiency of 5.3 miles/gallon. A 50% reduction in truck drag coefficient would give nearly 25% reduction in fuel consumption at highway speeds. This could save fuel worth \$1.5 billion per year [1]. Lower fuel consumption will also result in a reduction in pollution emissions and a reduced dependence on foreign oil.

A significant reduction in cost can be achieved by replacing some of the road and wind-tunnel testing required for the aerodynamic design and analysis of the Class 8 tractor-trailer with computational simulations. At the same time, one should view the computational results with caution because not all the flow physics may be completely captured by the computational fluid dynamics (CFD) simulations and the drag prediction may differ from the true values. The US Department of Energy has developed a multi-year project in which it has funded a number of organizations and universities with the aim to establish a clear understanding of the drag producing flow phenomena over a Class

8 tractor-trailer through joint experiments and computations. Eventually this research will help in the design of ‘smart’ drag reducing devices for this class of tractor-trailers [1]. One aim of this project is to provide high-quality aerodynamic data for direct validation of computational tools used for simulation of the flow over a Class 8 tractor-trailer. Use of the validated computational tools will help reduce the experimental cost in the future. As a first step of the validation study a very simplified model of a Class 8 tractor-trailer called the Ground Transportation System (Figure 2.1) was studied experimentally at NASA Ames [4]. A detail explanation of this experimental study is provided in Chapter 2. To perform validation studies, computational simulations of the Ground Transportation System (GTS) were performed using the Detached Eddy Simulation turbulence model [8]. The simulation approach is described in Chapter 3. Comparison with the experimental data and the results will be discussed in Chapters 4 and 5.

1.1 Research Objective

This work will compare the results from the computational simulations of the simplified tractor-trailer model (or the GTS model) with the experimental data to assess the Detached Eddy Simulation model as a CFD tool to simulate the flow over a Class 8 tractor-trailer. The comparison will include the drag coefficient of the flow over the truck surface, details of the flow structure in the wake of the truck, pressure coefficient distribution at various locations on the truck surface and the frequency of the vortices shed from the truck base.

1.2 Introduction to Turbulence Models

1.2.1 Reynolds Averaged Navier-Stokes Equations

From the engineering point of view we are usually interested in average or mean quantities while analyzing turbulent flows. Since turbulence can be characterized by random chaotic motion of the fluid, we rely on statistical approaches to calculate the averaged quantities. The Navier-Stokes equations are mass-averaged using a statistical approach called Favre (mass) averaging [6]. A Favre-averaged velocity vector is given by

$$\tilde{v}_i = \frac{1}{\bar{\rho}} \lim_{\Delta t \rightarrow \infty} \int_0^{\Delta t} \rho v_i dt \quad (1.1)$$

where $\tilde{v}_i$ is the Favre-averaged velocity vector, $\bar{\rho}$ is the mean density and v_i is the instantaneous velocity vector. So the Favre-averaged continuity equation (tensor notation) in conservation form is given by

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial(\bar{\rho} \tilde{v}_i)}{\partial s_i} = 0, \quad (1.2)$$

the Favre-averaged momentum equation in conservation form is given by

$$\frac{\partial(\bar{\rho} \tilde{v}_i)}{\partial t} + \frac{\partial(\bar{\rho} \tilde{v}_j \tilde{v}_i)}{\partial s_j} = -\frac{\partial \bar{p}}{\partial s_i} + \frac{\partial}{\partial s_j} \left[t_{ji}^- - \overline{\rho v_j'' v_i''} \right] \quad (1.3)$$

and the Favre-averaged energy equation in conservation form is given by

$$\begin{aligned}
\frac{\partial}{\partial t} \left[\bar{\rho} \left(\tilde{e} + \frac{\tilde{v}_i \tilde{v}_i}{2} \right) + \frac{\overline{\rho v_i'' v_i''}}{2} \right] + \frac{\partial}{\partial s_j} \left[\bar{\rho} \tilde{v}_j \left(\tilde{h} + \frac{\tilde{v}_i \tilde{v}_i}{2} \right) + \tilde{v}_j \frac{\overline{\rho v_i'' v_i''}}{2} \right] \\
= \frac{\partial}{\partial v_j} \left[-q_{Lj} - \overline{\rho v_j'' h''} + \overline{t_{ij} v_i''} - \frac{1}{2} \overline{\rho v_j'' v_i'' v_i''} \right] \\
+ \frac{\partial}{\partial s_j} \left[\tilde{v}_i \left(t_{ij}^- - \overline{\rho v_i'' v_j''} \right) \right]
\end{aligned} \tag{1.4}$$

Equations 1.2 to 1.4 are called the Reynolds-Averaged Navier-Stokes (RANS) equations [6]. It should be noted that they are obtained by performing Favre-averaging of the Navier-Stokes equations, but it is conventional to call them the RANS equations. The mean pressure is given by using the mean density and temperature in the perfect gas law

$$\bar{p} = \bar{\rho} R \tilde{T} \tag{1.5}$$

There are some unknown terms in the RANS equations. Any term in the RANS equations which involves the product of two or more Favre fluctuating variables (v_i'' and h'') is an unknown term because the exact correlation between two Favre fluctuating variables is not known. These terms are approximated by solving additional partial differential and algebraic equations. Depending on the number of partial differential equations used to approximate these unknown terms, the RANS models are divided into four categories [6].

Algebraic Models

The algebraic models use algebraic equations to solve for the unknown terms in the RANS. Some examples of these kind of models are the Cebeci-Smith model and the Baldwin-Lomax model [6].

One-Equation Models

One-equation models use one partial differential equation and some algebraic equations to approximate the unknown terms. Some examples of these kinds of models are the Spalart-Allmaras model and the Baldwin-Barth model [6].

Two-Equation Models

Two-equation models use two partial differential equations and some algebraic equations to approximate the unknown terms. Of all the RANS models two-equation models are the most popular for the industrial applications. The $k - \omega$ and $k - \epsilon$ models are the most popular two-equation models [6].

Second-Order Closure Models

The effects of stream line curvature, sudden changes in strain rate, secondary motion, etc. cannot be accommodated by two equation models. These features are handled by second order closure models. For three-dimensional flows, a second order closure model introduces seven new partial differential equations. Some examples of these kind

of models are the Launder-Reece-Rodi model and the Wilcox multiscale model. However, because of the large number of equations and complexity involved in second-order closure models they are not as popular as the two-equation model and the algebraic models [6].

1.2.2 Introduction to Large Eddy Simulation (LES)

The RANS equations model the effect of the turbulent eddies on the flow, but they do not resolve the turbulent eddies. Large Eddy Simulation (LES) is other turbulence modeling technique in which the larger eddies are resolved and the smaller ones are modeled. Resolving means the Navier-Stokes equations are numerically solved without any kind of approximation. LES is computationally expensive but is usually more accurate than RANS. Two important steps in LES are as follows [6]:

1. Filtering
2. Sub-grid scale modeling

Filtering separates the large eddies (which are resolved) from the smaller eddies (which are modeled). It introduces a scale Λ which represents the smallest turbulent scale or eddy that can be resolved by the LES model. Any scale or eddy smaller than Λ is modeled using a sub-grid scale model. Commonly used filters in LES are the box function, the Fourier cutoff function and the Gaussian function [6]. Some examples of sub-grid scale models are the Smagorinsky eddy viscosity model [41] and the Dynamic one-equation model [40].

1.2.3 Introduction to Hybrid RANS/LES Models

The biggest drawback of LES simulations for wall-bounded flows is that they are prohibitively expensive because of the very fine mesh requirements of the LES model in the boundary layer. To overcome this drawback the LES model is combined with a RANS model to give the hybrid RANS/LES modeling approach. In this approach RANS modeling is used in the attached boundary layer region and LES is used away from the boundary layer and in the highly separated flow regions [20]. Examples of the hybrid RANS/LES models are the Detached Eddy Simulation model (DES) [8] and the Linear Numerical Scale model (LNS) [7]. A detailed explanation of the DES model is given in Section 3.2.

1.3 Literature Review

In this section a review of some experimental and computational research performed on bluff body flows such as the GTS model, a square cylinder and a surface mounted cube is presented. The cases of the flow over a square cylinder and a surface mounted cube are lower Reynolds number cases and the case of the flow over the GTS model is a high Reynolds number case. This section will also compare the results from various CFD models used for modeling the flow over bluff bodies. Some new hybrid RANS/LES models are also reviewed.

1.3.1 Experimental Research

Cooper [2] has reviewed the past work done in the field of truck aerodynamics for drag reduction purpose. He has described the design aspects and the potential of drag reduction in the second generation drag reducing devices. Based on the data available he has concluded in his work that the combination of first and second generation devices can reduce fuel consumption by over 4,000 US gallons for a tractor-trailer operating 125,000 miles/year. A net reduction in wind-averaged drag coefficient based on a highway speed of 65 mph $\Delta\bar{C}_D(65) = 0.183$ is possible with a combination of first and second generation devices. The second generation devices are mechanically more complex than the first-generation aerodynamic package and their implementation will require efficient mechanical design and government action to encourage their use.

High Reynolds Number Flow Cases

The Reynolds number for a Class 8 tractor-trailer traveling on a highway at an average speed of 70 mph is approximately 6 million. So the flow over tractor-trailer is considered a high Reynolds number case. Effect of Reynolds number on the drag coefficient of a tractor-trailer is negligible for Reynolds number greater than 2 million.

McCallen et al. [1] have stressed the importance of the understanding of the flow structure in the wake and the flow in the gap between the tractor and the trailer for optimized design of the drag reducing devices. A brief summary of the suitability of various turbulence models for the computational simulations of the flow over the tractor-trailer

is discussed showing the limitations of the RANS models. Results from the experimental investigations of the drag reducing devices are discussed. It is concluded that these devices can provide up to 11% to 12% of fuel savings. The authors have mentioned the need to make the trailer manufacturers realize the potential of the drag reducing devices fitted on the trailers and a need to design these devices so that their maintenance cost is minimum.

In an experimental paper by Storms et al. [5] two studies were performed using the Generic Conventional Model (GCM). A GCM is a model of a Class 8 tractor-trailer with all the details of the shape of the tractor-trailer body.

1. The effect of variation of Reynolds number on the aerodynamic drag acting on the tractor-trailer.
2. The variation in the aerodynamic drag with use of various drag reducing devices.

In the first study it was found that the effect of the Reynolds number was most evident at high yaw angles ($\Psi > 8^\circ$ and $\Psi < -8^\circ$) where there was a significant reduction in drag at lower Reynolds numbers (Reynolds number was varied from 1 to 6.5 million). In the second study six drag reduction devices were studied experimentally. They are listed below.

1. Side and top extenders
2. Boattail plates
3. Base flaps

4. Trailer belly box
5. Trailer side skirts
6. Trailer full skirt

Table 1.1 shows the effectiveness of drag-reduction devices measured relative to the extender configuration. The positive sign indicates an increase in the wind averaged drag and the negative sign indicates a decrease in the drag value. The wind averaged drag reported in the paper was computed for a highway speed of 55 mph.

Table 1.1: Effect of drag-reduction concepts on wind-averaged drag [5].

Flow	ΔC_D , 55 mph	% difference
No Extenders	+0.156	+37.0
Extenders	0	0
Boattail plates	-0.058	-13.7
Base flaps	-0.082	-19.4
Trailer belly box	-0.050	-11.8
Trailer side skirts	-0.026	-6.2
Trailer full skirt	+0.016	+3.8

Low Reynolds Number Flow Cases

The low Reynolds number flow cases have the Reynolds number less than one million. The effect of wall proximity on flow in the wake of a square cylinder was analyzed by Martinuzzi et al. [10] at a Reynolds number of 18,900 based on the diameter. They observed four different flow regimes which are related to changes in the gap flow between the cylinder bottom face and the neighboring wall. For $b/D > 0.9$ the back pressure, the drag coefficient and the strength of the shed vortices in the wake of the cylinder are

insensitive to the gap height. For $0.6 < b/D < 0.9$, the strength of the shed vortices and fluctuating lift decrease while the base pressure increases as b/D is reduced. For $0.3 < b/D < 0.6$ the flow reattaches intermittently on the bottom faces, which causes the shedding of vortices to become increasingly irregular. For $b/D < 0.3$ the lower shear layer reattaches permanently to the bottom face and the periodic fluctuations are completely suppressed.

Martinuzzi and Tropea [11] studied the flow structures around obstacles with different aspect ratios placed in a flow channel. The Reynolds number based on the height of the channel was varied from 80,000 to 120,000. For aspect ratios $(w/H) > 6$ the cross-stream component of velocity and pressure gradient are negligible in the middle of the wake showing that this region can be treated as two-dimensional. It is shown that the total pressure loss due to the presence of the object is proportional to the blockage ratio. It is concluded that the recovery length behind a three dimensional obstacle is shorter than in the case of a two-dimensional flow.

Lyn et al. [12] studied experimentally the flow over a square cylinder at Reynolds number of $Re = 21,400$ based on the cylinder diameter. Ensemble-averaged statistics of the flow around the square cylinder are obtained using two component laser-Doppler measurement. A distinction is made between the flow in the near wake region where the vortices are ‘mature’ and have a distinct identity and the base region where the vortices grow to maturity and are then shed.

1.3.2 Computational Research

High Reynolds Number Flow Cases

Roy et al. [21] have performed RANS simulations on the GTS model at 0° yaw angle using the baseline Menter $k - \omega$ model. The quantities compared with the experimental data [4] include vehicle drag, surface pressure and the time-averaged velocities in the trailer near wake. The prediction of pressure and the time-averaged velocity in the trailer wake is in poor agreement with the experimental data due to the inability of the RANS models to accurately capture the near-wake vortical structure. This is one motivation for using the DES model (which actually resolves the larger eddies) to simulate the flow over the GTS model.

Maddox et al. [18] performed DES of the GTS model at 0° and 10° yaw angles and compared his results with experiment performed by Storms et al. [4]. The computational domain extended from approximately $15W$ upstream of the GTS up to $13.5W$ downstream of the GTS. The dimensionless time step (non-dimensionalized using the width W of the GTS model and upstream velocity 93.9 m/s) used was 0.02 . For the 10° yaw angle case the reattachment of the separation bubble on the leeward side of the model occurred substantially further along the GTS than indicated by the pressure measurements made by Storms [4]. The statistics were acquired for a non-dimensional time of 81 units. The present work does not agree with this time-window used by Maddox because from the research presented in this thesis the smallest time-window required for getting statistically converged results is $\Delta\tau_w \approx 180$ units. For the 0° yaw angle the

drag coefficient was calculated to be 0.279 as against the experimental value of 0.249. For the 10° yaw angle case the drag coefficient from the simulations was 1.379 and the experimental value was 1.253. No details of the wake flow have been shown or described. The authors have attributed the discrepancy in the force predictions at 10 degree yaw to the transition from laminar to turbulent flow in the boundary layers.

Unaune et al. [29] have performed DES on the simplified tractor-trailer at 0° yaw angle using commercial modeling software Fluent 6.1 and compared their results with the experiment performed by Storms et al. [4]. Unstructured mesh was used in the simulations. Wall functions were used to model the boundary layer and the first cell center was 0.75 mm from the truck surface. The grid consisted of 13.8 million cells. The time step used in the simulations were 0.001 s for first 0.08 s and 0.0005 s for the remaining simulation period. The estimated period of initial transients was 0.2 s and the statistics were collected for a time window of 0.2 s to 0.55 s. The research presented in this thesis does not agree with this and shows that the initial transience period is nearly equal to 1 s and the time-window required for obtaining statistically converged results is nearly equal to 0.6 s. The details of the wake given by Unaune show that the results are not statistically converged. The drag coefficient computed from the simulations was $C_D = 0.253$ while the experimental value of the drag coefficient is $C_D = 0.249$. In the power spectra of the unsteady pressure sensor located in the base of the trailer there were no obvious peaks showing any particular shedding frequency.

Baysal and Bayraktar [30] have used the renormalization group (RNG) $k - \epsilon$ model for the simulation of flow over a Generic Conventional Model (GCM). The computations were performed on the left half of the computational domain with proper boundary conditions at the symmetry plane. The fine mesh consisted of about 5 million cells. On the ground boundary the velocity of the flow was set equal to the flow at the inlet boundary to create a moving ground boundary condition. The flow over the tractor-trailer without tires and with stationary ground plane was also computed. The flow in the gap between the tractor and trailer was altered with removal of the tires. The computed drag value is about 6% less as a result of removing tires. For the stationary ground case the boundary layer on the ground thickened to alter the entire undercarriage flow. Also the trailer wake become skewed and was driven towards the ground. The computed drag value was 9% more as a result of making the ground stationary.

Low Reynolds Number Cases

Ramesh et al. [13] have performed 3D unsteady RANS simulations of the flow over a square cylinder at Reynolds number of $Re = 22,000$ based on the cylinder diameter. The RANS model used was a non-linear $k - \epsilon$ model [48]. Unlike the standard $k - \epsilon$ model the non-linear model uses non-linear terms in the calculation of Reynolds stresses. Although the results of the bulk parameters like the reattachment length, mean drag and the Strouhal number are inferior to results from LES simulations by Rodi [14] and by Nakayama and Vengadesan [15] and they do not produce same level of agreement with experimental data the prediction of the mean quantities and the unsteady phenomenon

is better than the linear eddy viscosity models. On the whole when comparing the computational time required by the present model and by the LES simulations, the accuracy achieved by the unsteady non-linear $k - \epsilon$ model is significant and hence this model can be used to simulate 3D unsteady complex engineering flows and reasonable success can be achieved in its application.

Roy et al. [20] have examined the flow over a square cylinder using steady state RANS methods and DES. The RANS modeling was performed on the vertical symmetry plane of the square cylinder. For the DES the numerical issues examined include statistical convergence, iterative convergence, temporal discretization error and spatial discretization error. All the RANS models greatly over predicted the length of the recirculation zone. The models gave poor predictions of the mean and fluctuating velocities in the wake. The results from the DES gave good agreement with the experimental data for the global quantities and the wake mean and rms velocities.

Rodi et al. [14] had organized a workshop on LES simulations of a square cylinder at a Reynolds number of $Re = 22,000$ and a surface mounted cube at Reynolds number of $Re = 3000$ and $Re = 22,000$. In the conclusion the authors have mentioned that the results from the simulations of the flow over a cube were in better agreement with each other and with the experimental data because the flow over a cube is fully turbulent, unlike the square cylinder case which involved transitional flow. The predictions produced by the use of different sub-grid scale models are different and the differences are larger at high Reynolds number. The dynamic one equation model gave better spatial distribution of

the sub-grid eddy viscosity, but there is no guarantee of getting overall better results relative to Smagorinsky model.

Lübcke et al. [22] have devised a RANS model called the explicit algebraic stress model (EASM) and applied it to the square cylinder at Reynolds number of $Re = 22,000$ and the circular cylinder case at a Reynolds number of $Re = 3900$ and $Re = 140,000$. The EASM model shows improvements in prediction of unsteady flows at low computational cost as compared to other RANS models. The results predicted by the EASM matched fairly well when compared with the LES simulations and the experimental data.

Krajnović and Davidson [23] performed LES simulations of the surface mounted cube at a Reynolds number of $Re = 40,000$. The sub-grid scale models used were the dynamic one equation model (OEM) proposed by Davidson [24] and the localized dynamic sub-grid scale kinetic energy equation model proposed by Menon and Kim [25]. A series of time-averaged velocities and turbulent stresses are computed and compared with the experimental data. The results of velocities obtained from the simulation are generally in better agreement with the experiment than the results obtained for the Reynolds stresses. Predictions made without any model give poor agreement with the experimental data but the two sub-grid models give good agreement with the experiment. It was observed that the grid refinement has greater effect on the separation length than the reattachment length. The Strouhal number is obtained from the Fourier transformation of the side force signal. The Strouhal number for the fine mesh using the OEM model was 0.146 and best agreed with the experimental result of 0.145. The authors concluded that LES

proposed in the paper using simple inlet boundary conditions, on a relatively coarse grid with the two sub-grid scale models gave accurate results at an acceptable computational cost.

Sohankar et al. [26] examined results from different sub-grid scale models used in LES simulations of flow around a square cylinder at Reynolds number of $Re = 22,000$. The sub-grid scale models used were the Smagorinsky model (SSM) [41], the standard dynamic model (DSM) [42] and the dynamic one-equation model [40]. When blockage was taken into account the DSM model gave the worst results when compared with the experiment. The SSM model produced similar results to those produced by the dynamic one-equation model except for the Strouhal number and the root mean square (rms) drag. Effects of spatial and temporal resolution and the computational span wise length were analyzed using the dynamic one-equation model. Finer spatial resolution improved agreement with the experiment. When the spanwise dimension was increased from four to seven diameters there was a 6% reduction in sectional rms drag. When the time resolution was increased by a factor of 2 the rms lift reduced by 5%. The one-equation sub-grid model was successful in accounting for the backscattering phenomenon. Among the three models the lowest level of all components of Reynolds stress is predicted by the DSM. It was concluded that lower Reynolds stresses correspond to higher pressure region in the wake flow which leads to lower drag force. The results produced by the dynamic one-equation model give better agreement with experiment than the other two subgrid models.

Barone and Roy [28] used the DES model for simulation of the bluff body wake flow. The authors have used a combination of Yee's [32] second order symmetric total variation diminishing (STVD) scheme along with characteristic-based filtering for solving the Navier-Stokes equations. The characteristic-based filtering was obtained using the artificial compression method (ACM) switch of Harten [33]. The first case analyzed was flow in the wake of a square cylinder at a Reynolds number of $Re = 21,400$. Out of the coarse, medium and fine grids, results from the fine grid are mostly superior than other grids, except the rms drag fluctuation and prediction of the stream-wise velocity fluctuation in the near wake region. The DES model with the present numerical scheme is not able to give accurate predictions of Reynolds shear stress in near wake. In the summary for this case the fine grid DES results are found competitive with LES calculations in the prediction of global quantities. The results of the coarse mesh using the low dissipation scheme (combination of STVD and ACM switch) gave nearly same results as the fine mesh using the STVD scheme. It is concluded that the DES model succeeds where RANS models often fail in predicting the mean flow and global flow quantities.

Rodi [31] has used RANS and LES to calculate flow over bluff bodies and the results are compared with the experimental data. The bluff bodies used are a long square cylinder ($Re = 22,000$) and a surface mounted cube ($Re = 40,000$). Calculations were carried out using $k - \epsilon$ model. $k - \epsilon$ model with the Kato Launder [34] correction was also used. The Kato Launder correction eliminates spurious turbulence production from

the $k - \epsilon$ model in the stagnation region. A two-layer approach was used in connection with $k - \epsilon$ model for both cases. In the two-layer approach [35] the viscous sub-layer is resolved with a simpler one-equation model. Smagorinsky's SGS model and Germano's dynamic model were used as sub-grid scale models.

For the square cylinder, results provided by the $k - \epsilon$ model are poor. The Kato-Launder modification yields improved results. Further improvement in the results is obtained with the combination of the two-layer approach resolving the near wall region and the Kato-Launder correction. In all RANS calculations the turbulent fluctuations are severely under-predicted. It was found that none of the LES results are uniformly good and entirely satisfactory, and there were large differences between the individual calculations which are difficult to explain. For the surface mounted cube the $k - \epsilon$ model leads to poor prediction over the top. The result is improved with Kato-Launder correction. A combination of the two layer approach and the Kato-Launder correction gives better results for the global quantities except the separation length which worsens when we shift from standard model to the combination. LES gives better results overall because it simulates all the complex features of the 3-dimensional flow. The difference between various LES calculations is much smaller for the surface mounted cube when compared with those from the square cylinder.

Hybrid RANS/LES Models

Spalart and Squires [16] have addressed the precautions which should be taken to obtain better results from simulation of flow over bluff bodies using the DES model. They are listed below.

1. Very long time samples are essential to converge the statistics, as vortex shedding has strong modulations in DES. Simulating only a few cycles of such shedding is unsafe.
2. The authors have mentioned that the error resulting from dissipation from numerical discretization of the Navier-Stokes equations combined with the DES model can be large and requires great care in controlling it. One of the solution to this problem is increasing the grid density by refining the grid and varying the time step involved in the simulations.
3. A precautionary check should be performed by the user to confirm that the RANS/LES interface is not deep inside the boundary layer because it leads to inaccuracies in the solution.

The authors have also highlighted some areas of research regarding the use of DES model that need to be investigated.

1. Research on the flexible design of DES simulation which will involve LES treatment within the boundary layer particularly upstream of the separation line because this has given better results in some cases [17].

2. Another issue is the area in which a shear layer, after separation, needs to generate “LES content” which it did not possess as a boundary layer upstream. If the grid is not constructed properly the DES model may not create any LES content. This issue may be important for the research presented in this thesis because shear layers separated from the boundary layers formed on the GTS surfaces are released in the LES region and are not correctly simulated.
3. A source of ambiguity exists in use of DES for vortex-dominated flows because of the effect of the DES limiter. The risk involved is that the solution may not have LES behavior in the intended LES region.

Nichols [27] used three hybrid RANS/LES models to simulate the flow over a circular cylinder ($Re = 8 \times 10^6$). The models were:

1. Spalart-Allmaras DES model (SA-DES model)
2. Shear Stress Transport DES model (SST DES model)
3. SST-Multiscale DES model

The SST DES model and SST-Multiscale model transition from RANS to LES as a function of local turbulent length scale and the local grid spacing unlike the Spalart-Allmaras DES model. The Spalart-Allmaras DES model transitions from RANS to LES as a function of local grid spacing only. The SA-DES model did not reach grid convergence for the cylinder average drag in the refinement study made by the author. The SST-based models are less sensitive to grid density variations than the SA-DES

model. The grid sensitivity of the SA-DES model indicates that a filter function that includes some measure of the local turbulence length scales along with the local grid scale is superior to a filter function that includes only the local grid scale. A computational mesh with a grid resolution resulting in a ratio of the turbulent length scale to grid length scale greater than two produced a reasonable simulation with all three hybrid models for the square cylinder case.

Camarri et al. [7] have devised a hybrid RANS/LES model called as the Limited Numerical Scale (LNS) model. They have applied it to the simulation of the flow around a square cylinder at a Reynolds number of $Re = 22,000$. The model uses local blending of two eddy viscosities. One eddy viscosity comes from Smagorinsky's sub-grid scale model in the LES and other comes from the $k - \epsilon$ RANS model. In LNS, unlike DES, there is no a-priori separation between regions to be treated with LES and RANS models. Also, DES is based on the Spalart-Allmaras RANS model while the LNS strategy is directly applicable to most available RANS models. Simulations carried out on a grid sufficiently refined for LES show that the LNS results are almost identical to those given by LES. For a coarser resolution the results from LNS are better than LES and RANS because of its blending criteria.

CHAPTER 2

PROBLEM DESCRIPTION

2.1 Wind Tunnel Experiment

A 1/8-scale simplified model of the tractor-trailer geometry (or the GTS model) was studied experimentally in the NASA Ames 7 ft×10 ft wind tunnel. A complete set of the experimental data and the surface definition of the model are included on a CD-ROM for further analysis and comparison. The wind tunnel has 4.57 m long test section and a constant height of 2.13 m. The width of the test section is 3.05 m with 1% wall divergence [4]. Figure 2.1 shows the GTS geometry used during the experiment. The length of the model is 2.48 m ($x/W = 7.647$), width $W = 0.324$ m and the height is 0.45 m ($y/W = 1.392$). This model is without any wheels and there is no tractor-trailer gap, which simplifies the grid generation. The dimensions of the wind tunnel and the GTS model were non-dimensionalized by the trailer width $W = 32.38$ cm. Because of the interest in wake details, the GTS model was located 13.33 cm downstream of the beginning of the test section of the wind tunnel and the bottom of the model was located 7.6 cm above the wind-tunnel floor. A body-axis coordinate system was used in the experiment. Force and pressure measurements were made at yaw angles (Ψ) ranging from -14° to 14° . Yaw angle is the angle made by the longitudinal axis of a vehicle with the direction of the air flow. Detailed data sets were obtained at Reynolds numbers of $Re = 2 \times 10^6$ and $Re = 7.4 \times 10^5$. These Reynolds numbers are based on the width of the trailer. These numbers

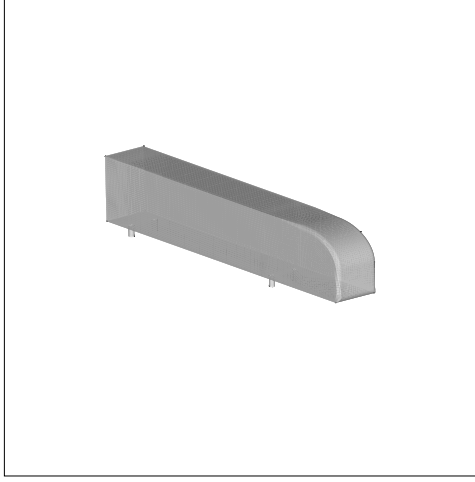


Figure 2.1: The GTS model

correspond to wind-tunnel velocities of 205 mph and 75 mph respectively [4]. It should be noted that the computational simulations are carried out in this research are only for the case of Reynolds number of $Re = 2 \times 10^6$ and a yaw angle of $\Psi = 0^\circ$. Table 2.1 shows the wind tunnel flow conditions for the $Re = 2$ million experiment.

Table 2.1: Wind tunnel conditions [4].

$\Psi, ^\circ$	0
Re , million	2.021
M	0.279
V_∞ , m/s	94.48
q , N/m ²	5266.82
p_t , N/m ²	102492.48
p_s or p_{ref} , N/m ²	97216.07
T_t , K	285.76
T_s , or T_{ref} , K	276.59
$\bar{\rho}$, kg/m ³	1.180

2.1.1 Surface Pressures

79 surface pressure taps were located on the model and 43 additional pressure taps were located on the right wall of the test section (looking upstream). These pressure taps were used to study the pressure distribution around the GTS model. In addition, one unsteady pressure transducer was located on the rear of the trailer ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$). The unsteady pressure signal was used to obtain power spectra of the unsteady pressure. The reference static pressure tap was located at $x/W = 4.47$, $y/W = 2.588$ and $z/W = -4.7$. This location is on the wall of the wind tunnel. See figures in [4] for the location of the pressure taps. The reference static pressure is used in calculation of the pressure coefficient and the drag coefficient. The uncertainty in the pressure coefficient calculated from the experiment was ± 0.002 . This uncertainty estimate includes only the measurement precision adjusted for the run-to-run repeatability [4]. Table 2.2 shows the experimental pressure coefficient distribution along the y/W axis of the trailer base at three z/W locations. These values are available in the CD-ROM. The values of these experimental pressure coefficients will be used for comparison with computational results.

Table 2.2: Experimental values of pressure coefficient at different z/W locations along the y/W axis of the trailer base [4].

y/W location	C_p at $z/W = 0$	C_p at $z/W = 0.2206$	C_p at $z/W = 0.44120$
1.3333	-0.080177	-0.087359	-0.082658
1.0147	-0.101592	-0.107468	-0.104073
0.6961	-0.162834	-0.162051	-0.153302
0.3775	-0.202139	-0.198483	-0.187775
0.0588	-0.155130	-0.152519	-0.143639

2.1.2 Drag, Lift and Side Force Measurements

The drag measurements are accurate to within ± 1 lbf (4.45 N). The repeatability of the drag coefficient measurements was ± 0.001 for $\Psi \geq -5^\circ$ and ± 0.01 for $\Psi < -5^\circ$. These error bands include measurement resolution and point-to-point repeatability [4]. The accuracy of the side and lift forces is not mentioned. The experimental values of drag, lift and side force coefficient will be compared with simulation results in Section 5.

2.1.3 Oil-film Interferometry (OFI), Hot-film Anemometry and Particle Image Velocimetry (PIV)

Oil-film interferometry was done on the top and right side of the GTS model to measure the skin friction on those surfaces. The uncertainty in this technique is $\pm 5\%$. Hot-film measurements were made on the right side of the cab to determine the existence and extent of flow separation in the vicinity of the front corner radius. Three-component PIV was conducted in several planes in the wake of the trailer to measure the velocities in those planes [4].

CHAPTER 3

SIMULATION APPROACH

3.1 Governing Equations

The Favre-averaged Navier-Stokes equations (equations 1.2 to 1.4) are solved in the simulations. The Mach number at which the experiment was performed was $M = 0.279$ and the solver used for the simulations was a compressible fluid flow solver. Equations 1.2 to 1.4 are in differential form. It should be noted that the differential and the integral forms of the Navier-Stokes equations are equivalent, and the CFD solver used in this research uses integral form of the Navier-Stokes equations (i.e., the finite-volume method).

3.2 The DES Model

The DES model used in this work is a hybrid RANS/LES model proposed by Spalart and co-workers [8]. It uses the Spalart-Allmaras one-equation model for modeling the attached boundary layer region and LES for the simulations of highly separated flow regions. The Spalart-Allmaras model [6] uses one partial differential equation for calculation of the working variable $\tilde{\nu}$

$$\frac{\partial \tilde{\nu}}{\partial t} + v_i \frac{\partial \tilde{\nu}}{\partial s_i} = c_{b1} \tilde{S} \tilde{\nu} - c_{w1} f_w \left(\frac{\tilde{\nu}}{d} \right)^2 + \frac{1}{\sigma} \frac{\partial}{\partial s_i} \left((\nu + \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial s_i} \right) + \frac{c_{b2}}{\sigma} \frac{\partial \tilde{\nu}}{\partial s_i} \frac{\partial \tilde{\nu}}{\partial s_i} \quad (3.1)$$

The working variable is used for the calculation of the eddy viscosity

$$\nu_t = \tilde{\nu} f_{v1} \quad (3.2)$$

Other than the main partial differential equation used for calculating the working variable the model uses following auxiliary equations

$$f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3} \quad (3.3)$$

$$f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \quad (3.4)$$

$$f_w = g \left(\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right)^{1/6} \quad (3.5)$$

$$\chi = \frac{\tilde{\nu}}{\nu} \quad (3.6)$$

$$g = r + c_{w2} (r^6 - r) \quad (3.7)$$

$$r = \frac{\tilde{\nu}}{\tilde{S}_\kappa^2 d^2} \quad (3.8)$$

$$\tilde{S} = S + f_{v2} \frac{\tilde{\nu}}{\kappa^2 d^2} \quad (3.9)$$

$$S = \sqrt{2\Theta_{ij}\Theta_{ij}} \quad (3.10)$$

The tensor Θ_{ij} is the rotation tensor and is given by

$$\Theta_{ij} = \frac{1}{2} \left(\frac{\partial v_i}{\partial x_j} - \frac{\partial v_j}{\partial x_i} \right) \quad (3.11)$$

The closure coefficients [6] are given by

$$\begin{aligned} c_{b1} &= 0.1355 \\ c_{b2} &= 0.622 \\ c_{v1} &= 7.1 \\ \sigma &= \frac{2}{3} \\ c_{w1} &= \frac{c_{b1}}{\kappa^2} + \frac{(1 + c_{b2})}{\sigma} \\ c_{w2} &= 0.3 \\ c_{w3} &= 2 \\ \kappa &= 0.41 \end{aligned} \quad (3.12)$$

The transition from RANS modeling in the boundary layer to LES outside the boundary layer is performed by changing the definition of the distance d of a point to the nearest

wall in the equation 3.9 [8]. The distance d is replaced with $\tilde{d}$.

$$\tilde{d} = \min(d, C_{DES}\Delta) \quad (3.13)$$

The constant C_{DES} is given by Spalart [8] and is equal to 0.65. Far from the wall, the value of $\tilde{d}$ becomes

$$\tilde{d} = C_{DES}\Delta \quad (3.14)$$

where Δ is the local grid spacing and is equal to maximum mesh spacing in the three coordinate directions

$$\Delta = \max(\Delta x, \Delta y, \Delta z) \quad (3.15)$$

When the production term is balanced with the dissipation term we get the dynamic eddy viscosity

$$\mu_t = \bar{\rho} (C_{Sm}\Delta)^2 \sqrt{S_{ij}S_{ij}} \quad (3.16)$$

and the value of the constant C_{Sm}

$$C_{Sm} = C_{DES} \sqrt{\frac{c_{b1}f_{v1}}{c_{w1}f_w}} \quad (3.17)$$

where S_{ij} is the strain rate tensor

$$S_{ij} = \frac{1}{2} \left(\frac{\partial \bar{v}_i}{\partial x_j} + \frac{\partial \bar{v}_j}{\partial x_i} \right) \quad (3.18)$$

In the outer part of the boundary layer C_{Sm} asymptotes to $C_{Sm} = 0.29C_{DES} = 0.19$. The DES model thus asymptotes to a Smagorinsky-type LES model in the bluff-body wake assuming sufficient mesh refinement [8].

3.3 The SACCARA Code

The SACCARA (Sandia Advanced Code for Compressible Aerothermodynamics Research and Analysis) code is a compressible fluid flow solver at the Sandia Laboratories and was used for the DES computations performed during this research. It was developed from the INCA code [37] written by Amtec Engineering. It is used to solve the Navier-Stokes equations and the turbulence transport equations in conservation form. Code verification has been performed on the SACCARA code by code-to-code comparison with other Navier-Stokes codes [38] and with Direct Simulation Monte Carlo method [39]. The code uses a Lower-Upper Symmetric Gauss-Seidel scheme to solve the equations. This scheme is based on the works of Yoon et al. [44] and Peery and Imlay [46] and has excellent scalability up to thousands of processors.

3.4 Discretization of the Convective Terms

3.4.1 Finite Volume Method

The discretization approach used for convective terms is the finite volume method (FVM). The FVM relies on obtaining the governing equations in integral form. It can also be used for unstructured grids unlike the finite difference methods. The SACCARA code is based on cell-centered finite-volume approach. The cell-centered finite-volume approach involves evaluation of exact integrals of properties over a cell by approximating them by the values at the center of the cell.

3.4.2 Upwind Schemes

An upwind scheme was used for the discretization of the convective terms. In the upwind scheme the convective term can be expressed as a central difference term plus a numerical diffusion term. These methods provide a physically-based method for introducing numerical diffusion. They can capture sharp gradients/discontinuities and they minimize dispersion. Three types of upwind schemes are the flux vector splitting (FVS) approach, the flux difference splitting (FDS) approach and hybrid upwind schemes. The SACCARA code has available both FVS and FDS schemes, and a FDS scheme was used for the DES.

The current simulations employ Yee's symmetric total variation diminishing (TVD) scheme. The scheme is modified to incorporate Harten's artificial compression switch (ACM) [33] to further reduce the dissipation, along with a characteristic based filtering

method [32]. Yee's scheme uses the Roe interface flux, which may be written as

$$Z_{m+1/2} = \frac{Z_m + Z_{m+1}}{2} + \frac{1}{2} R1_{m+1/2} \Phi_{m+1/2} \quad (3.19)$$

Here $\frac{1}{2} [Z_m + Z_{m+1}]$ is the central differencing portion of the numerical flux $Z_{m+1/2}$ and $R1_{m+1/2} \Phi_{m+1/2}$ is the nonlinear dissipation term. $R1_{m+1/2}$ is the right eigenvector of the matrix $\partial Z_{m+1/2} / \partial U_{j+1/2}$ where U is the conservation flow vector. The non-linear dissipation term is combined with Harten's switch and this switch is used to decide the amount of non-linear dissipation in a non-dissipative scheme. The modified dissipation term looks like

$$Z_{m+1/2}^* = \frac{1}{2} R1_{m+1/2} \Phi_{m+1/2}^* \quad (3.20)$$

where the elements of $\Phi_{m+1/2}^*$ are given by

$$\phi_{m+1/2}^{l*} = K \theta_{m+1/2}^l \phi_{m+1/2}^l \quad (3.21)$$

The $\phi_{m+1/2}^l$ are the elements of the vector $\Phi_{m+1/2}$ in the symmetric TVD scheme and are given by

$$\phi_{m+1/2}^l = - \left| \lambda_{m+1/2}^l \right| \left(\alpha_{m+1/2}^l - Q_{m+1/2}^l \right) \quad (3.22)$$

where $\lambda_{m+1/2}^l$ is the eigenvalue of $\partial Z_{m+1/2}/\partial U_{j+1/2}$ and $Q_{m+1/2}^l$ is the minmod limiter. The dissipation can be scaled down globally by reducing the value of the numerical dissipation constant K . The value of K varies between 0 to 1. $\theta_{m+1/2}^l$ serves as Harten's switch and it can be used to further scale down the numerical dissipation in regions of smooth flow

$$\theta_{m+1/2}^l = \frac{|\alpha_{m+1/2}^l - \alpha_{m-1/2}^l|}{|\alpha_{m+1/2}^l + \alpha_{m-1/2}^l|} \quad (3.23)$$

where $\alpha_{m+1/2}^l$ are the elements of $R1_{j+1/2}^{-1}(U_{m+1} - U_m)$.

3.5 Discretization of the Diffusion Terms

The viscous terms were discretized using central differencing.

3.6 Temporal Discretization

Sub-iterative procedure is used to obtain second-order accuracy during temporal discretization. In this the summation of the discretized temporal derivative and the steady-state residual at the $n + 1$ time level is iterated until it reaches the specified tolerance value.

3.7 Grid Generation Approach

The grid generation approach used during this research was the structure grid approach using the commercial grid generation software Gridgen. In this approach the grid

is laid in a regular repeating pattern called block. For a 3-dimensional mesh the structured grid uses hexahedral elements in a computationally rectangular array. The grid can be stretched and twisted to fit it according to the body shape. The multi-block grid generation approach which is used in this research allows several blocks to be connected together to construct the whole domain. The requirements of block to block connectivity can vary according to the software used. The SACCARA code uses point to point connectivity where the blocks must match topologically and physically at the boundary.

Structured grids provide a very high degree of control over the grid but the time and expertise required to create a structured grid for the entire model is a challenging task [49].

3.8 Domain Decomposition and Load Balancing

In dealing with geometrically large complicated systems the mesh domain can be decomposed into subdomains and each subdomain solved on a separate processor of a cluster or a supercomputer. This method is called domain decomposition method (DDM). The main advantages of DDM include efficiency of solvers, savings in computational storage conducive to parallel processing and saving in computational time [45].

In a DDM method the domain $\Omega(t)$ is expressed as a union of subdomains

$$\Omega(t) = \bigcup_{c=1}^{n_p(t)} \Omega_c(t) \quad (3.24)$$

An important consideration while solving large grids on distributed memory architectures is the distribution of the mesh across the memory of the machine at runtime so that the calculated load is evenly balanced and the amount of interprocessor communication is minimized [45]. For the present case the coarse and medium meshes were sub-divided into 98 sub-domains, and these sub-domains were distributed over 86 processors for both grids. Table 3.1 shows the number of cells in coarse and medium mesh the number processors used for the simulations and the average number of cells/processor.

Table 3.1: Coarse and medium mesh

Mesh	Number of cells	n_p	average number of cells/processor
Coarse	3.9×10^6	86	45,349
Medium	1.32×10^7	86	153,488

3.9 Boundary Conditions used for the Simulations

The inflow boundary employs stagnation values for pressure $p_t = 102,653 \text{ N/m}^2$ and temperature $T_t = 282.1 \text{ K}$ and enforces inflow normal to the boundary. The outflow boundary used a fixed static pressure of $p_s = 97,700 \text{ N/m}^2$. The back pressure was chosen so that the tunnel wall reference pressure located at $x/W = 4.47$, $y/W = 2.588$ and $z/W = -4.7$ matched with the experiment. The reference pressure from the experiment and the average reference pressure from the coarse mesh simulations are compared in figure 3.1. The experimental reference pressure is $97,336 \text{ N/m}^2$ and the average reference pressure from the coarse mesh simulations is $97,295 \text{ N/m}^2$. It can be seen in the figure that the reference pressure from the simulations matches pretty well with reference pressure from

the experiment. Table 3.2 compares average reference pressure values from the coarse and medium mesh with the experimental value. Slip conditions on velocity are employed on the top and side walls of the wind tunnel to save the computational cost, while the floor of the wind tunnel, the GTS surface and the support posts employ no-slip velocity conditions and assume an adiabatic wall. The freestream dynamic eddy viscosity is set at $\mu_t = 1 \times 10^{-5}$ Ns/m². Solid wall boundary condition for the turbulence model can be found in [47].

Table 3.2: Reference pressure comparison

Experiment	Coarse Mesh	Medium Mesh
97,336 N/m ²	97,295 N/m ²	97,249 N/m ²

3.10 Characteristic Scales

Time has been non-dimensionalized by a reference time scale defined as

$$\tau = W/V_\infty = 0.0034s \quad (3.25)$$

Where W is the trailer width and V is the freestream velocity. The value of V_∞ was calculated from the reference static pressure and stagnation pressure and it came out to be 93.8 m/s¹. Characteristic length scales for the medium and coarse meshes are given in Table 3.3 (Note that the length scales are non-dimensionalized by the trailer width). Extremely fine grid spacing is required near the wall because the RANS model is integrated to the wall. The maximum y^+ value near the wall from the coarse mesh

is 1.4. The maximum grid spacing in the trailer wake leads to approximately 26 points across the trailer width for the coarse grid case and 38 points for the medium grid case. Figures 3.2 and 3.3 show the coarse and the medium mesh structure near the truck base. The medium mesh has higher grid density than the coarse mesh.

Table 3.3: Characteristics scales

Mesh	Δ_{wall}	Δ_{wake}
Coarse	6.17×10^{-5}	0.12
Medium	3.08×10^{-5}	0.081

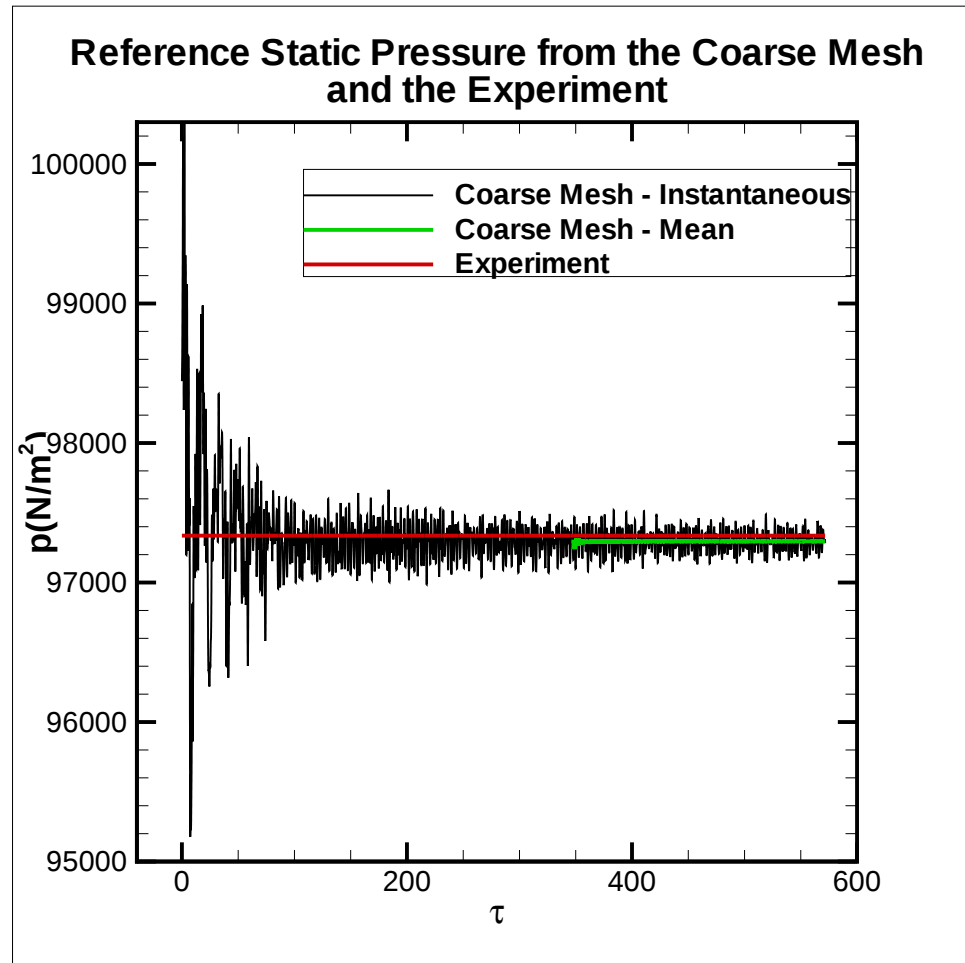


Figure 3.1: Reference static pressure comparison between the coarse mesh simulations and the experiment

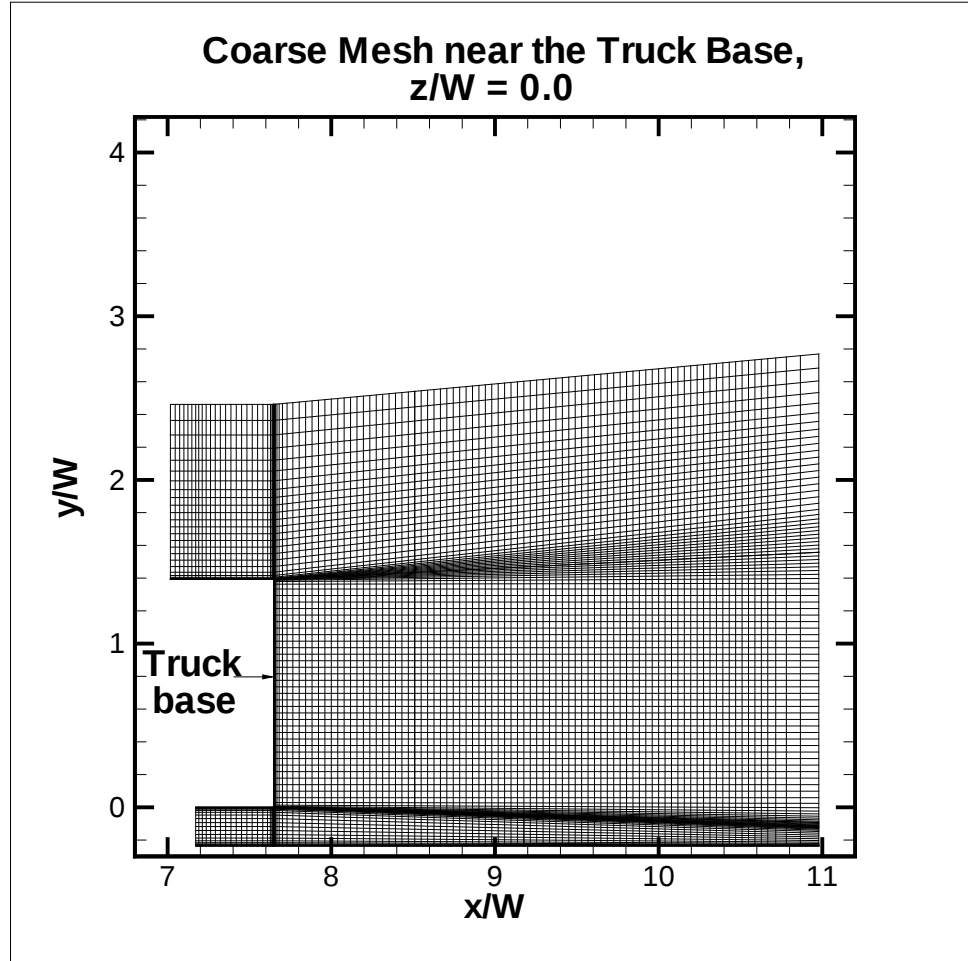


Figure 3.2: Coarse mesh structure near the truck base

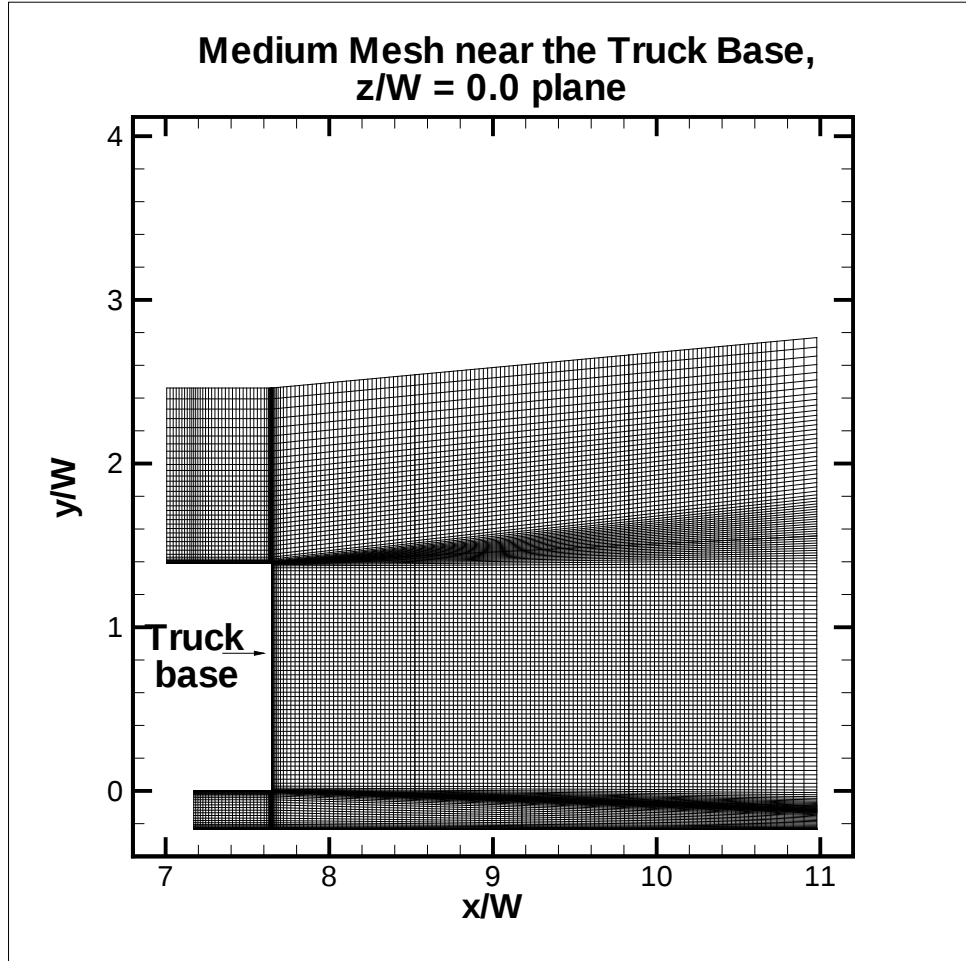


Figure 3.3: Medium mesh structure near the truck base

CHAPTER 4

NUMERICAL ACCURACY

Numerical accuracy is concerned with difference between numerical solution and exact solution to a differential equation. The Navier-Stokes equations for general compressible flows do not have an exact solution. In this Chapter we will assess the numerical accuracy of the solution by comparing it with the best available solution. In Section 4.1 an effort is made to predict the length of time window required for obtaining statistically converged results. In Section 4.2, the time period of existence of initial transients in the solution is estimated. In Section 4.3, the results are analyzed to find out the number of sub-iterations required for obtaining numerically accurate results. In Section 4.4 effect of the temporal discretization on the results is studied. In Section 4.5, the effect of the mesh size on the solution is studied. It should be noted that discretization error cannot be fully assessed in this case because PDEs solved in the DES model change with a change in the mesh size. Hence the whole model is changed when it is applied from the coarse to the medium mesh.

4.1 Statistical Convergence

DES is time dependent and to assess mean properties from these simulations we need to average the data from the simulations over some time window. The required size of the time window is determined by performing a statistical convergence analysis.

In this research the pressure and velocities in the base of the truck were used to perform this analysis. The reason for selecting these variables near the base region was that this is the unsteady flow region where the eddies are formed and shed. The flow over the front, top, side and bottom of the GTS model is mostly steady flow without any massive separation. Hence this flow is modeled using RANS. The pressure and velocity data were time averaged over a successively increasing time window. When the profiles from two successive time windows matched each other within a specific tolerance then the smaller time window out of the two windows is said to be the time window over which the data should be averaged to get statistically converged results. Figure 4.1 shows the unsteady pressure signal in the base of the truck ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$) plotted against non-dimensional time τ . The width of the rectangular windows correspond to the successively increasing time windows used for averaging the pressure and velocities. Note that the averaging starts in the reverse direction along the time-axis starting from $\tau = 571$. This is done in order to stay away from the initial transient period. In figure 4.2 the line T which is parallel to the y/W axis shows the location along which the pressure profiles are time-averaged. Line A , which is parallel to x/W axis, shows the location along which the velocity profiles are time averaged to obtain statistical convergence.

Figure 4.3 shows the the pressure coefficient (C_p) distribution plotted along the line T and corresponding to the various time windows shown in figure 4.1. In the figure it can be seen that only the C_p profile corresponding to $\tau_{start} = 527$ or the profile

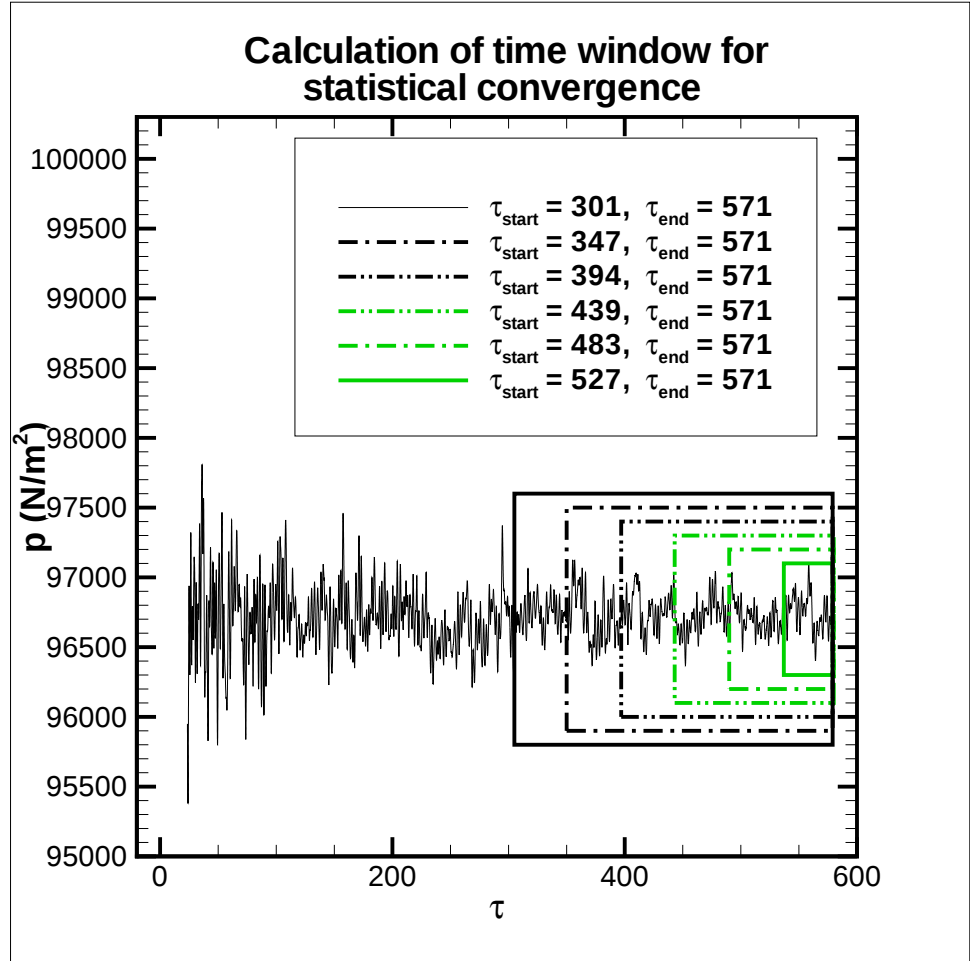


Figure 4.1: Time windows used for calculating statistically converged results

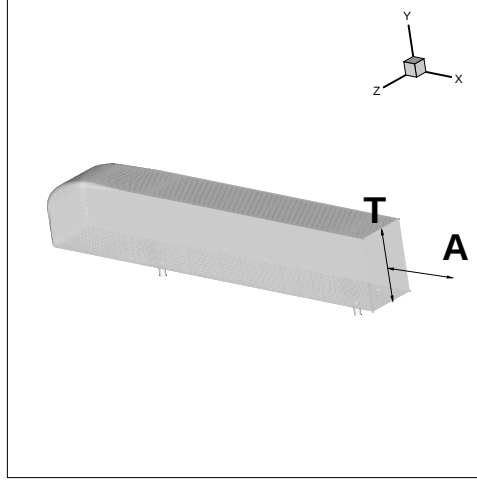


Figure 4.2: Location of pressure and velocity profiles used for statistical convergence

corresponding to smallest time window is not statistically converged. All other profiles seem to be statistically converged. Figure 4.4 shows the u/V_{ref} , v/V_{ref} and u_{rms}/V_{ref} velocity profiles plotted along the line A and corresponding to the various time windows shown in figure 4.1. In this figure a look at the v/V_{ref} and u/V_{ref} profiles suggests that all the profiles except the one corresponding to $\tau_{start} = 527$ seem to be statistically converged. But the u/V_{ref} profiles corresponding to the three biggest time windows are different from three profiles corresponding to the three smaller windows at some places. Figure 4.5 shows the absolute errors in all the C_p profiles corresponding to the various time windows shown in figure 4.1 with respect to the C_p profile corresponding to the biggest time window in figure 4.1. The biggest time window gives the best statistically converged results because it is averaged over longest period of time. Figures 4.6, 4.7 and 4.8 show the absolute errors in the u/V_{ref} , v/V_{ref} and u_{rms}/V_{ref} velocity profiles,

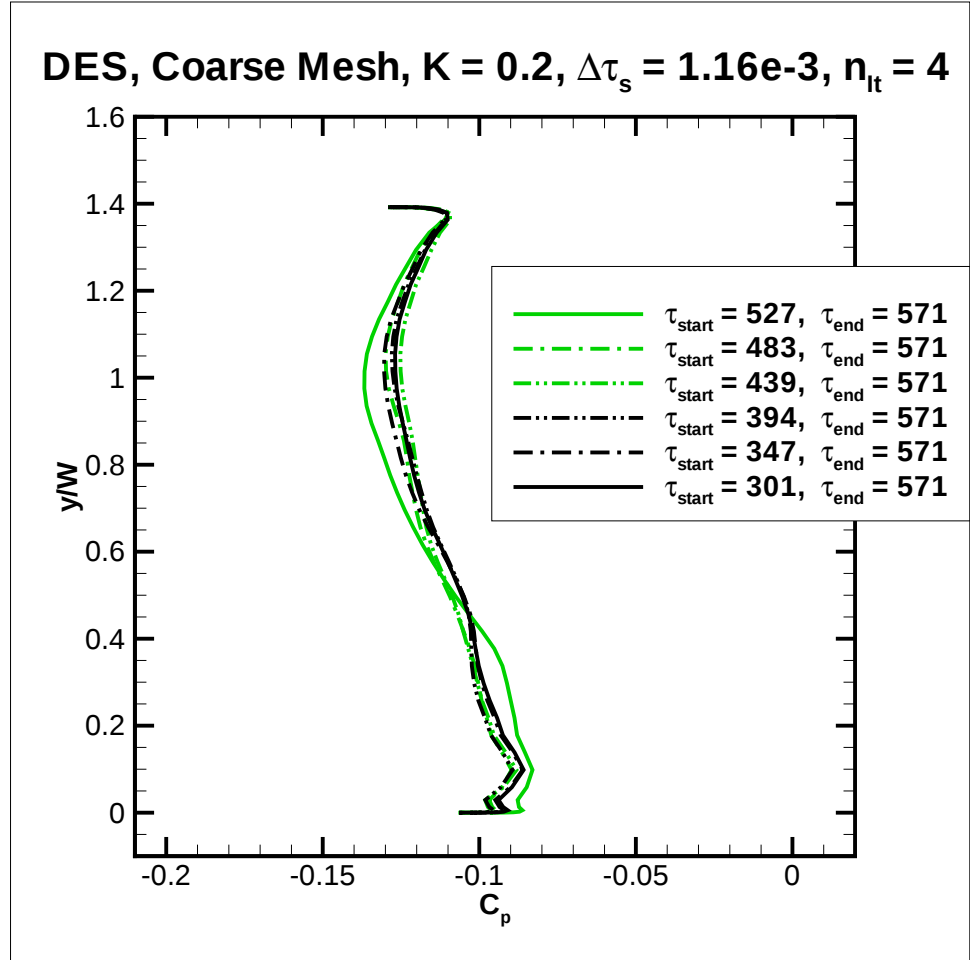


Figure 4.3: Pressure distribution along line T and corresponding to different time windows

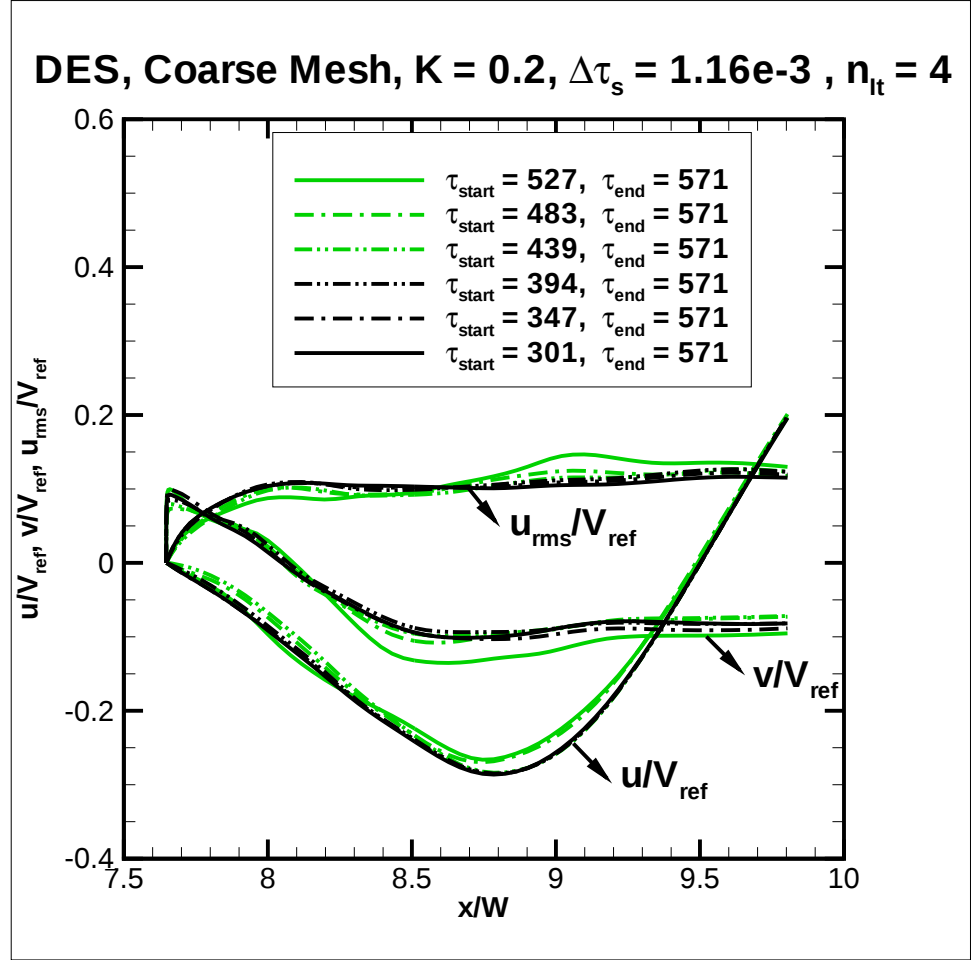


Figure 4.4: Velocity profiles along line A and corresponding to different time windows

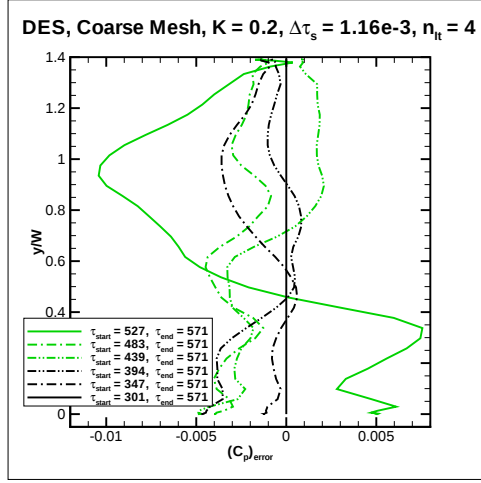


Figure 4.5: Error in the pressure distribution obtained from smaller time windows with respect to the pressure distribution obtained from the biggest time window

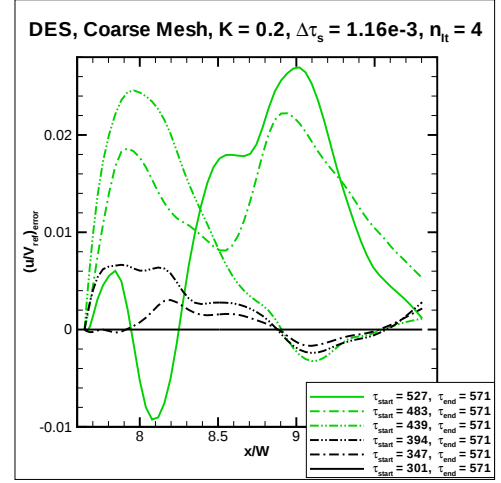


Figure 4.6: Error in the u/V_{ref} velocity profiles obtained from smaller time windows with respect to the u/V_{ref} velocity profile obtained from biggest time window

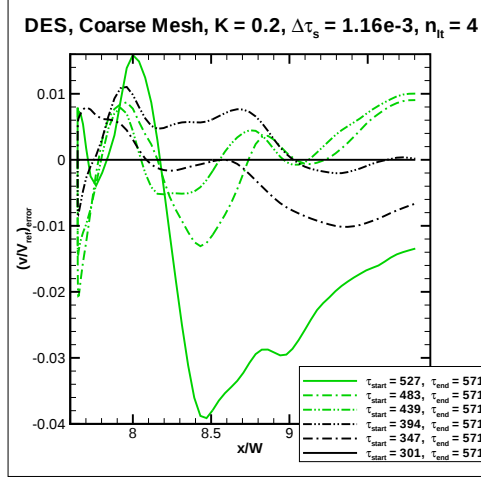


Figure 4.7: Error in the v/V_{ref} velocity profiles obtained from smaller time windows with respect to the v/V_{ref} velocity profile obtained from biggest time window

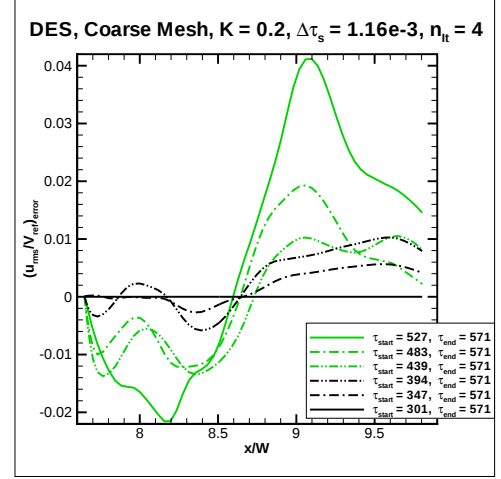


Figure 4.8: Error in the u_{rms}/V_{ref} velocity profiles obtained from smaller time windows with respect to the u_{rms}/V_{ref} velocity profile obtained from biggest time window

respectively, corresponding to the various time windows shown in figure 4.1 with respect to the biggest time window. Table 4.1 shows the largest absolute errors with respect to the biggest time window for various profiles seen in figures 4.5, 4.6, 4.7 and 4.8 (the relative percentage error is not calculated because the denominator in some cases may be zero). From the plots it is concluded that the smallest time window showing statistically converged results is the window with $\Delta\tau_w = 177$ because it has reasonably small errors with respect to the biggest time window. Other observation made from the various plots and the table is that the C_p profiles converges the fastest and the v/V_{ref} profile tends to converge the slowest. Although this analysis is based on coarse mesh results, it is assumed to hold true for the medium mesh results as well. Henceforth, any analysis done with the coarse and medium mesh is by averaging the data over time $\Delta\tau_w = 177$ for achieving statistically converged results. This non-dimensional time window corresponds to a physical time of $\Delta t_w = 0.611$ s.

Table 4.1: Absolute error with respect to biggest time window

$(C_p)_{error}$	$(u/V_{ref})_{error}$	$(v/V_{ref})_{error}$	$(u_{rms}/V_{ref})_{error}$	$\Delta\tau_w$
0.01041	0.02694	0.03916	0.04119	44
0.00448	0.0222	0.02063	0.019286	88
0.00488	0.02457	0.01425	0.01342	132
0.00470	0.00665	0.01103	0.01025	177
0.00359	0.00303	0.01015	0.00557	224
0	0	0	0	270

4.2 Initial Transients

Since the simulations are initialized with freestream values for velocity pressure and temperature the flow over the GTS during some initial time period oscillates because of the waves generated due to sudden exposure of the truck to the freestream flow. Figure 4.9 shows the plot of the unsteady pressure signal from the truck base region plotted against time. The large fluctuations in the pressure signal near $\tau = 0$ denotes the period of initial transients. As the simulations are reduced in time the initial transients die out. The data collected during the initial transient period should be neglected, thus the time period of existence of initial transients needs to be calculated. Figure 4.9 shows the time window of width $\Delta\tau_w \approx 177$ (this width was obtained from statistical convergence analysis in section 4.1) moved along the time-axis. Figures 4.10 and 4.11 show the time-averaged pressure and velocity profiles corresponding to the time windows shown in figure 4.9. The C_p profiles seen in 4.10 show that the profiles corresponding to first four time windows (starting from $\tau = 0$) are in the initial transience period. The last three pressure profiles seem to overlap each other and it can be said that initial transients have minimized in this period. Almost the same conclusion can be drawn after observing the time averaged u/V_{ref} , v/V_{ref} and u_{rms}/V_{ref} velocity profiles in figure 4.11.

Figure 4.12 shows the absolute errors in the C_p profiles corresponding to various time windows shown in figure 4.9 with respect to the C_p profile corresponding to the time window starting at $\tau_{start} = 394$ (or the final time window). The final time window is furthest along the time-axis from the initial transience period hence it is considered

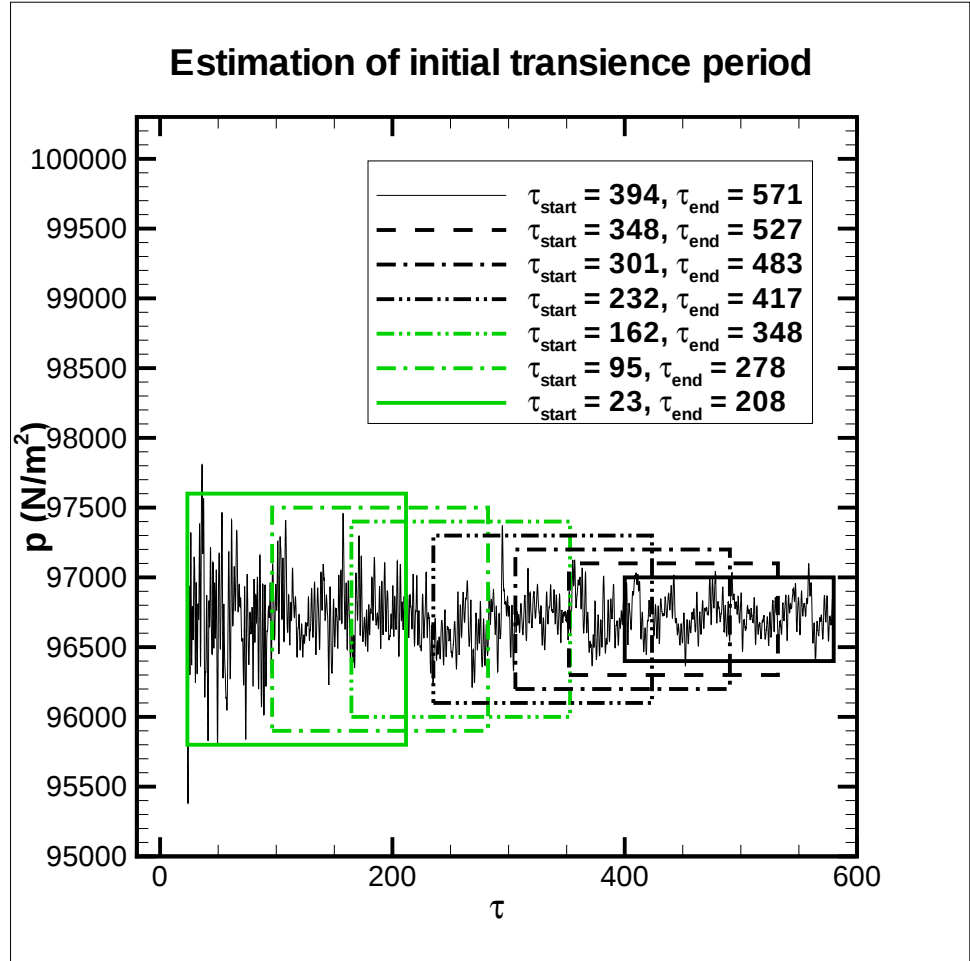


Figure 4.9: Time windows used for estimating period of initial transients

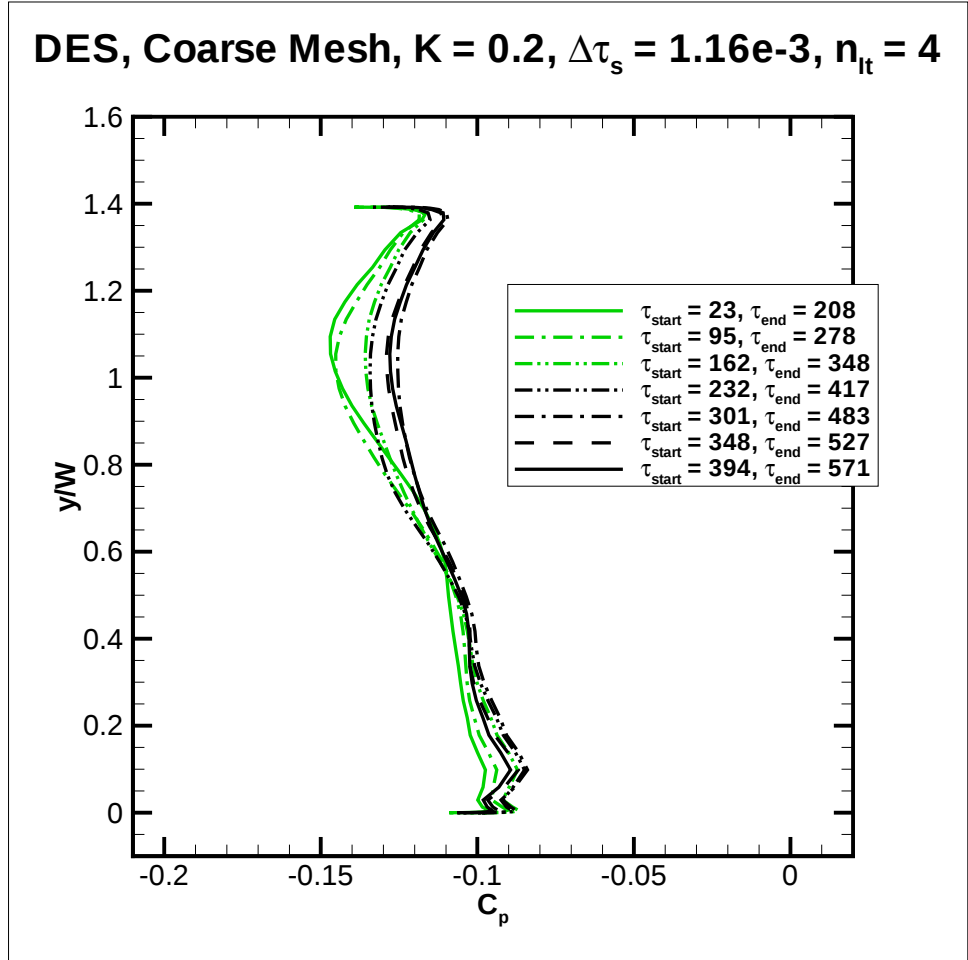


Figure 4.10: Pressure distribution along line T and corresponding to different time windows

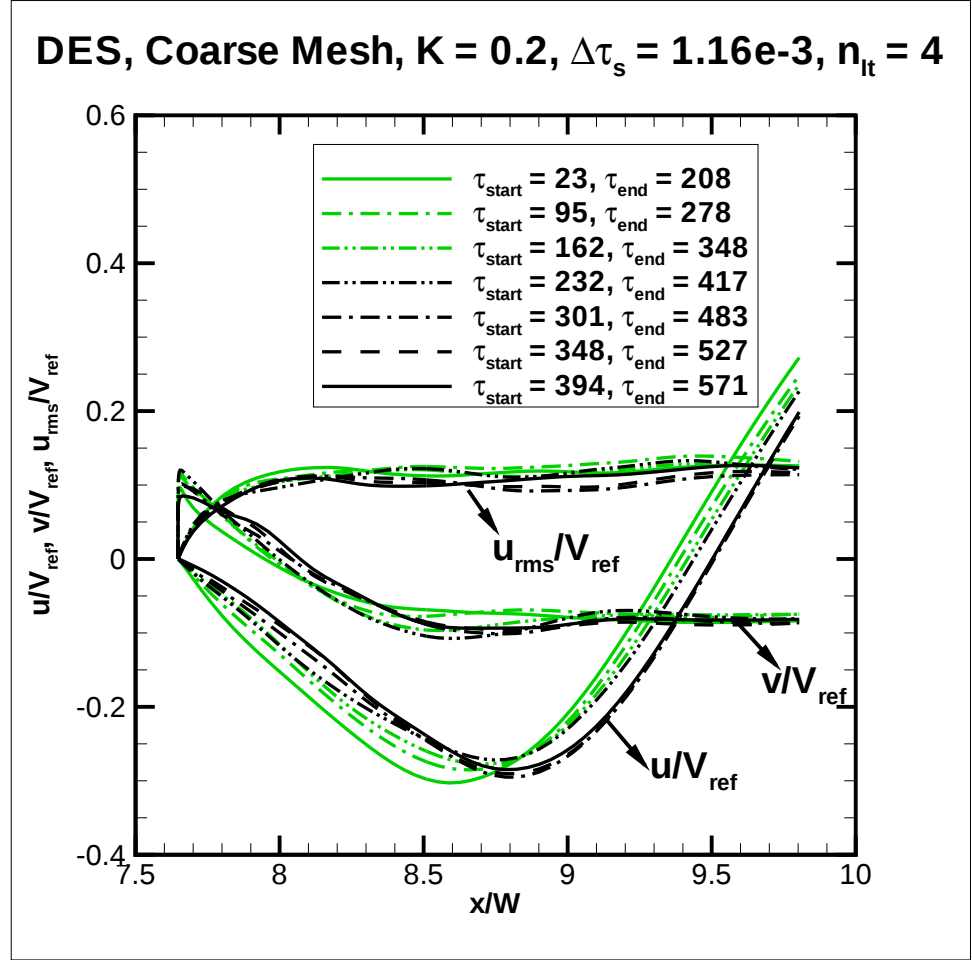


Figure 4.11: Velocity profiles along line A and corresponding to different time windows

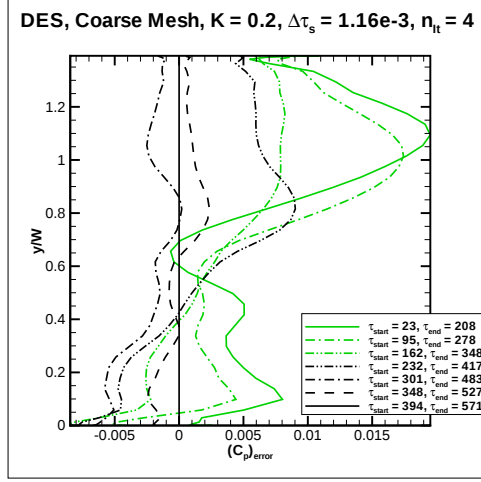


Figure 4.12: Error in the pressure distribution from all time windows with respect to the pressure distribution obtained from the window farthest along time-axis

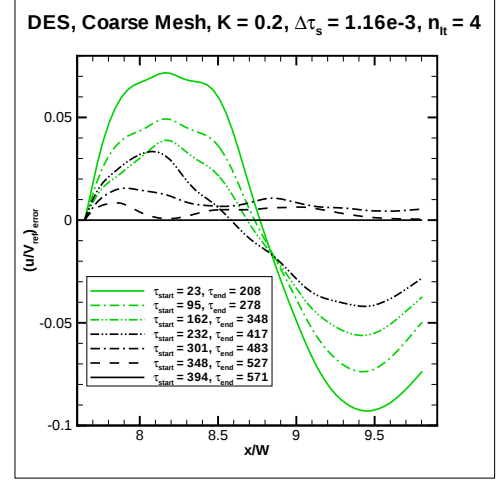


Figure 4.13: Error in the u/V_{ref} profiles from all time windows with respect to the u/V_{ref} profile obtained from the window farthest along time-axis

the ideal window. Figures 4.13 to 4.15 show the absolute errors in the u/V_{ref} , v/V_{ref} and u_{rms}/V_{ref} profiles corresponding to all the time windows shown in figure 4.9 with respect to the final time window. It is very obvious that the pressure and velocity profiles corresponding to window starting at $\tau_{start} = 23$ will show maximum errors because of the initial transients present in the results. Table 4.2 shows the biggest absolute errors in the pressure and velocity profiles from all the windows with respect to pressure and velocity profiles from the window corresponding to $\tau_{start} = 394$ (or the final window). It is observed from the various plots and the table that the C_p profile converges the fastest and the u_{rms}/V_{ref} profile converges the slowest. This analysis helps in estimating the period of initial transients. Examination of the pressure and velocity errors from table 4.2 suggests that the errors corresponding to windows with $\tau_{start} = 301$ and $\tau_{start} = 348$

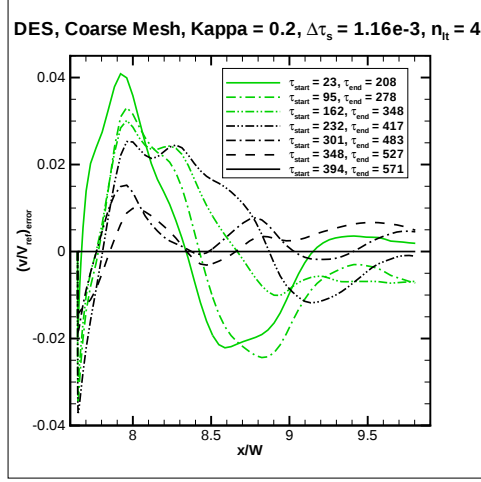


Figure 4.14: Error in the v/V_{ref} profiles from all time windows with respect to the v/V_{ref} profile obtained from the window farthest along time-axis

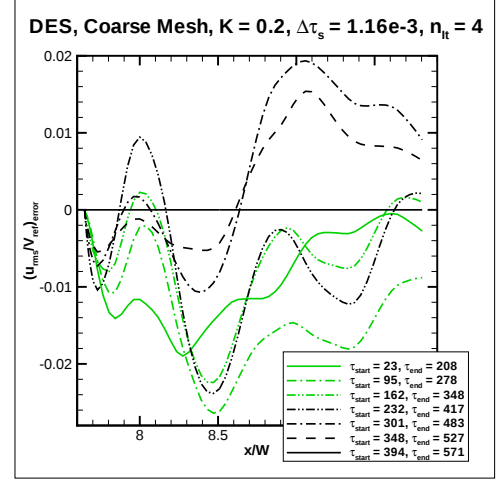


Figure 4.15: Error in the u_{rms}/V_{ref} profiles from all time windows with respect to the u_{rms}/V_{ref} profile obtained from the window farthest along time-axis

are acceptable and match closely with the profiles corresponding to the window with $\tau_{start} = 394$ (or the final window). Hence we conclude that the initial transients period exists from $\tau = 0$ to $\tau = 301$. Therefore any data in the region from $\tau = 0$ to $\tau = 301$ will be neglected. This corresponds to a physical time period of $\Delta t_w \approx 1$ s. All the comparisons with the experimental data will be made by using the data obtained after $\tau = 301$.

Table 4.2: Absolute error with respect to time window farthest along the time-axis

Time window	$(C_p)_{error}$	$(u/V_{ref})_{error}$	$(v/V_{ref})_{error}$	$(u_{rms}/V_{ref})_{error}$
$\tau_{start} = 23, \tau_{end} = 208$	0.01951	0.09295	0.04087	0.01897
$\tau_{start} = 95, \tau_{end} = 278$	0.01743	0.07368	0.03312	0.02639
$\tau_{start} = 162, \tau_{end} = 348$	0.00822	0.05606	0.03014	0.02241
$\tau_{start} = 232, \tau_{end} = 417$	0.00902	0.04203	0.02519	0.02383
$\tau_{start} = 301, \tau_{end} = 483$	0.00246	0.01557	0.01524	0.01934
$\tau_{start} = 348, \tau_{end} = 527$	0.00664	0.00850	0.00992	0.01533
$\tau_{start} = 394, \tau_{end} = 571$	0	0	0	0

4.3 Effect of Number of Sub-iterations used for Iterative Convergence

The number of sub-iterations to be performed for achieving iterative convergence during each time step is not exactly known for this complicated turbulent flow. The time steps used for the simulations are small ($\Delta\tau_{s1} = 1.162 \times 10^{-3}$ and $\Delta\tau_{s2} = 2.325 \times 10^{-3}$) and for such a small time steps, four to six sub-iterations performed for each time-step is believed to give an iteratively converged solution. The effect of the number of sub-iterations performed during each time-step was studied by performing one, four and eight sub-iterations per time step and the pressure and velocity profiles from these cases are compared in figures 4.16 and 4.17. The comparison of the pressure profile shows that the profiles corresponding to $n_{It} = 4$ and $n_{It} = 1$ are equidistant from the n_{It2} profile, which should be the most accurate profile because of the larger number of sub-iterations. Hence no conclusion can be made from the pressure plot. The effect of number of sub-iterations is clearly visible while comparing the u/V_{ref} velocity profiles. Comparison of the velocity profiles in figure 4.17 shows that the u/V_{ref} profile corresponding to $n_{It} = 1$ shows the biggest absolute error of approximately 0.03 with respect to the $n_{It} = 8$ case. The u/V_{ref} velocity profile corresponding to $n_{It} = 4$ shows much smaller error with respect to the $n_{It} = 8$ case. It is concluded from the velocity profile plot that four sub-iterations are sufficient for achieving iterative convergence when the time step of $\Delta\tau_{s1} = 1.162 \times 10^{-3}$ is used in the simulations.

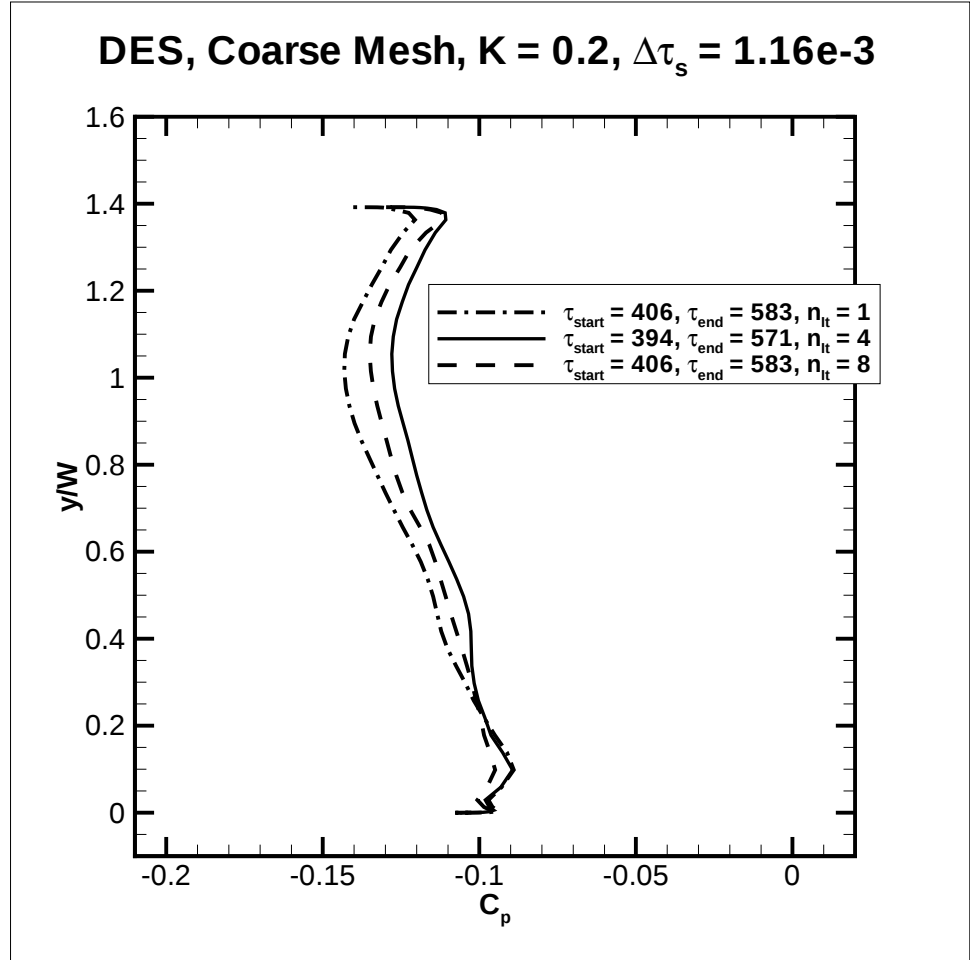


Figure 4.16: Pressure distribution along line T and corresponding to different number of sub-iterations

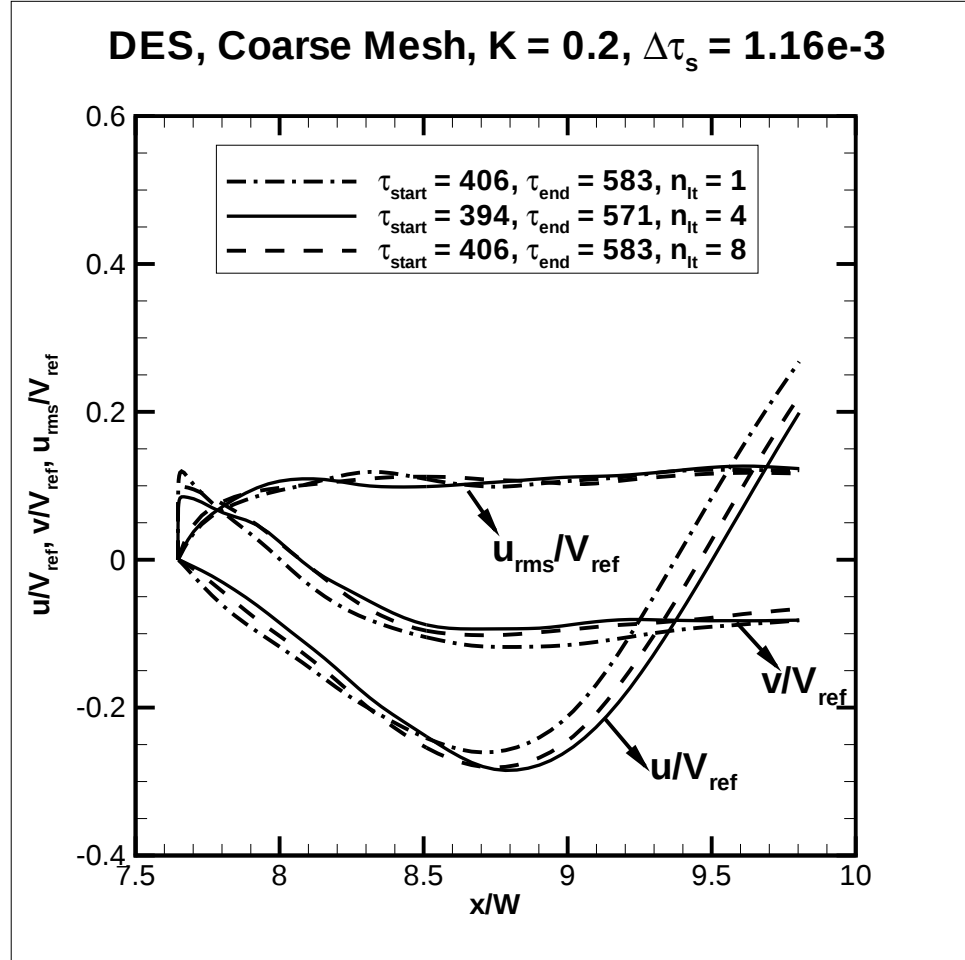


Figure 4.17: Velocity profiles along line A and corresponding to different number of sub-iterations

4.4 Effect of Time Step ($\Delta\tau_s$)

The effect of time step ($\Delta\tau_s$) used for discretizing the Navier-Stokes equations was analyzed by using two time steps for the coarse mesh. As mentioned previously the time-steps used were $\Delta\tau_{s1} = 1.162 \times 10^{-3}$ ($\Delta t_{s1} = 4 \times 10^{-6}$ s) and $\Delta\tau_{s2} = 2.325 \times 10^{-2}$ ($\Delta t_{s2} = 8 \times 10^{-6}$ s). Figure 4.18 shows the pressure distribution at the $z/W = 0$ along the vertical axis of the truck base (Line T in Figure 4.2) obtained from the two time-steps used. Figure 4.19 shows the velocity profiles (along line A in Figure 4.2) of the u , v and u_{rms} velocity profile obtained from the two time steps used for discretization. There is a significant difference in the pressure and velocity profiles obtained from the two time steps. One has to be cautious while comparing the profiles from these two time steps. The number of sub-iterations used for iterative convergence at the time step with $\Delta\tau_{s1}$ case is 4 and for the $\Delta\tau_{s2}$ case is 6. As mentioned in section 4.3 the number of sub-iterations that need to be performed at each time step to guarantee iterative convergence of the solution is not known. So even after performing 6 sub-iterations for the $\Delta\tau_{s2}$ case it is not known that the solution iteratively converged to the same level as in the $\Delta\tau_{s1}$ case with 4 sub-iterations. So it cannot be concluded whether the difference between the pressure and velocity profiles corresponding to the two time steps used is a result of temporal discretization error or an error due to iterative convergence. We can conclude that the difference between the pressure and velocity profiles from the two time-steps used for the simulations is a combination of the error due to iterative convergence, error due temporal discretization and presence of some transients in the pressure and velocity

profiles for the $\Delta\tau_{s2}$ case. The presence of transients in the $\Delta\tau_{s2}$ case comes into the picture because this case was obtained by changing the time step from $\Delta\tau_{s1}$ to $\Delta\tau_{s2}$ at time $\tau = 394$ during the simulations.

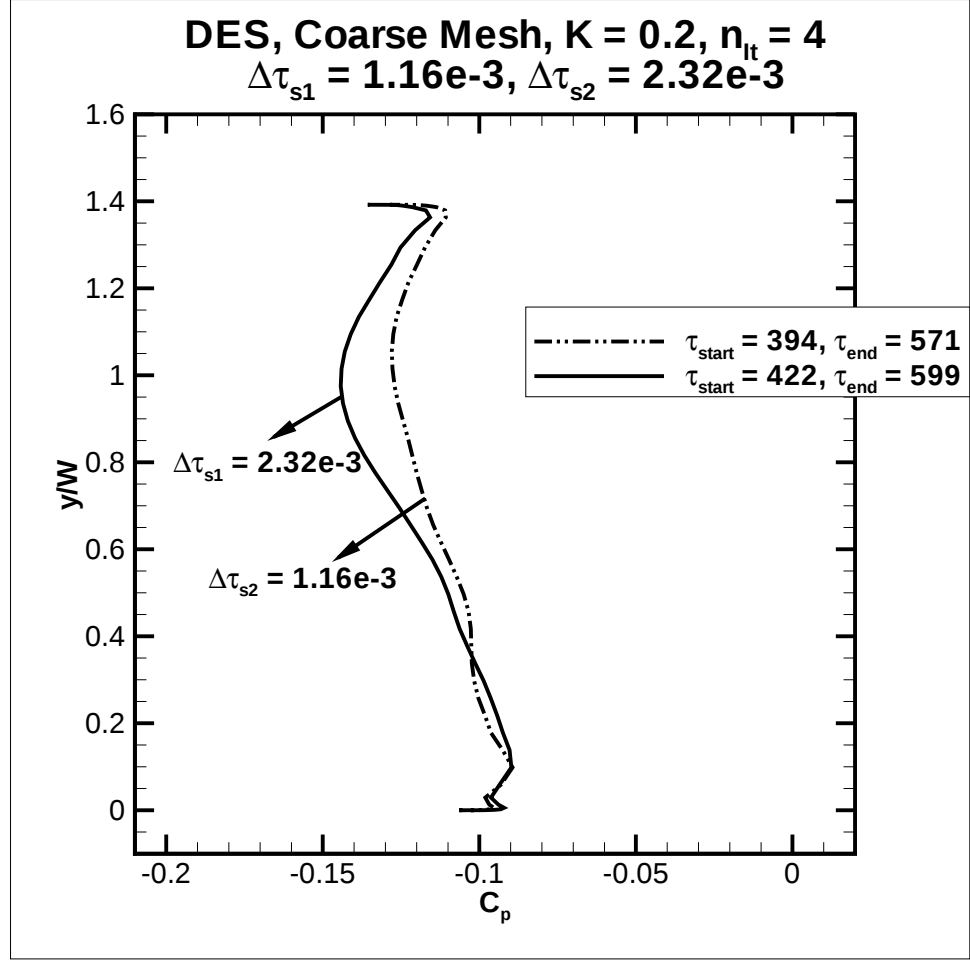


Figure 4.18: Pressure distribution along line T and corresponding to different time-steps

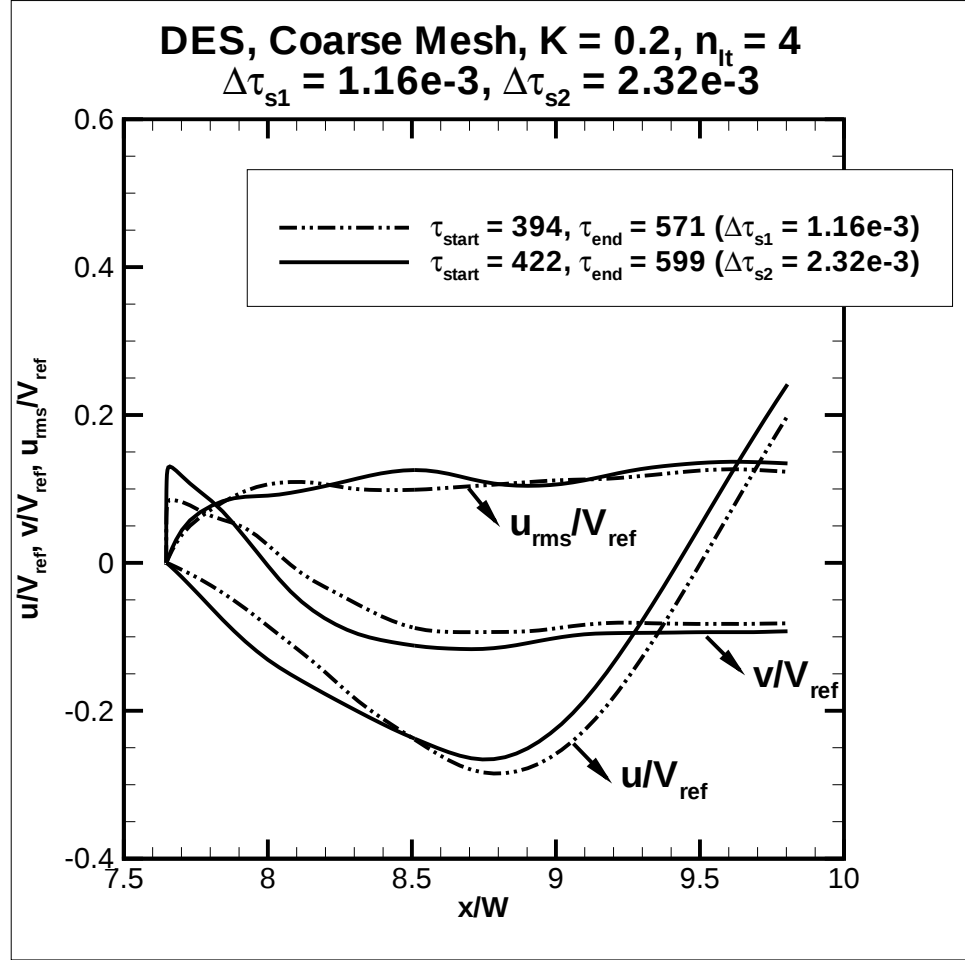


Figure 4.19: Velocity profiles along line A and corresponding to different time-steps

4.5 Effect of the Mesh Size on DES

The results from the medium and coarse mesh are compared in this section to examine the effect of the mesh size on the results from the simulations. Figure 4.20 and 4.21 show the u/V_{ref} velocity contours and streamlines in the vertical streamwise plane (red colored plane in figure 5.1) from the medium and the coarse mesh simulations. The upper vortex from the coarse mesh simulations is much more dominant than the lower vortex, while the vortices are almost equal in size for the medium mesh results. Figure 4.22 compares the pressure distribution obtained from the two grids. The pressure distribution obtained from the medium mesh is much more flat than the pressure distribution from the coarse mesh. The reason for this is the almost equal dominance of the upper and lower vortices in the medium mesh results in figure 4.20 and the larger size of the upper vortex in the coarse mesh simulations in figure 4.21. Figure 4.23 compares the velocity profiles from the two grids. The u/V_{ref} and u_{rms}/V_{ref} velocity profiles from the grids match to each other but the v/V_{ref} profiles from the two grids are different. The reason for this can again be attributed to the difference in size of the upper vortex obtained from the coarse and medium grid results.

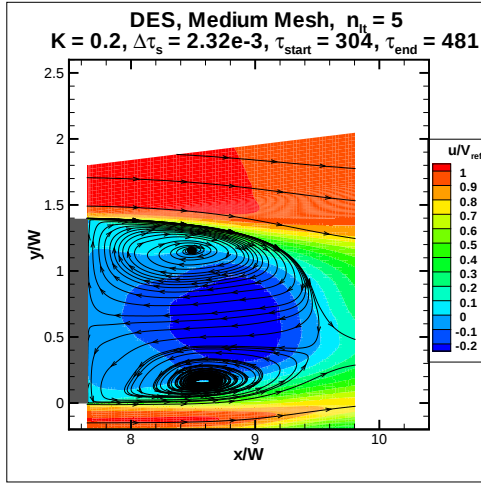


Figure 4.20: Medium mesh vertical stream-wise contour plot

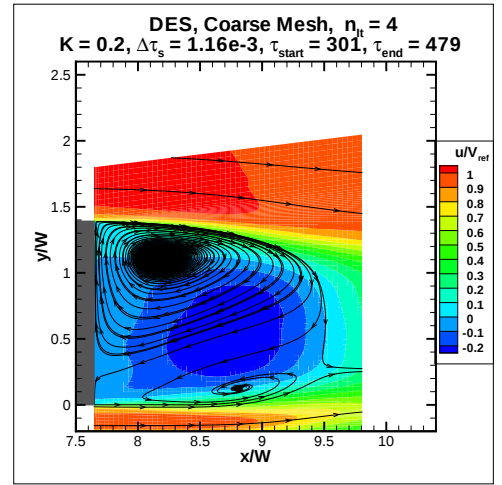


Figure 4.21: Coarse mesh vertical stream-wise contour plot

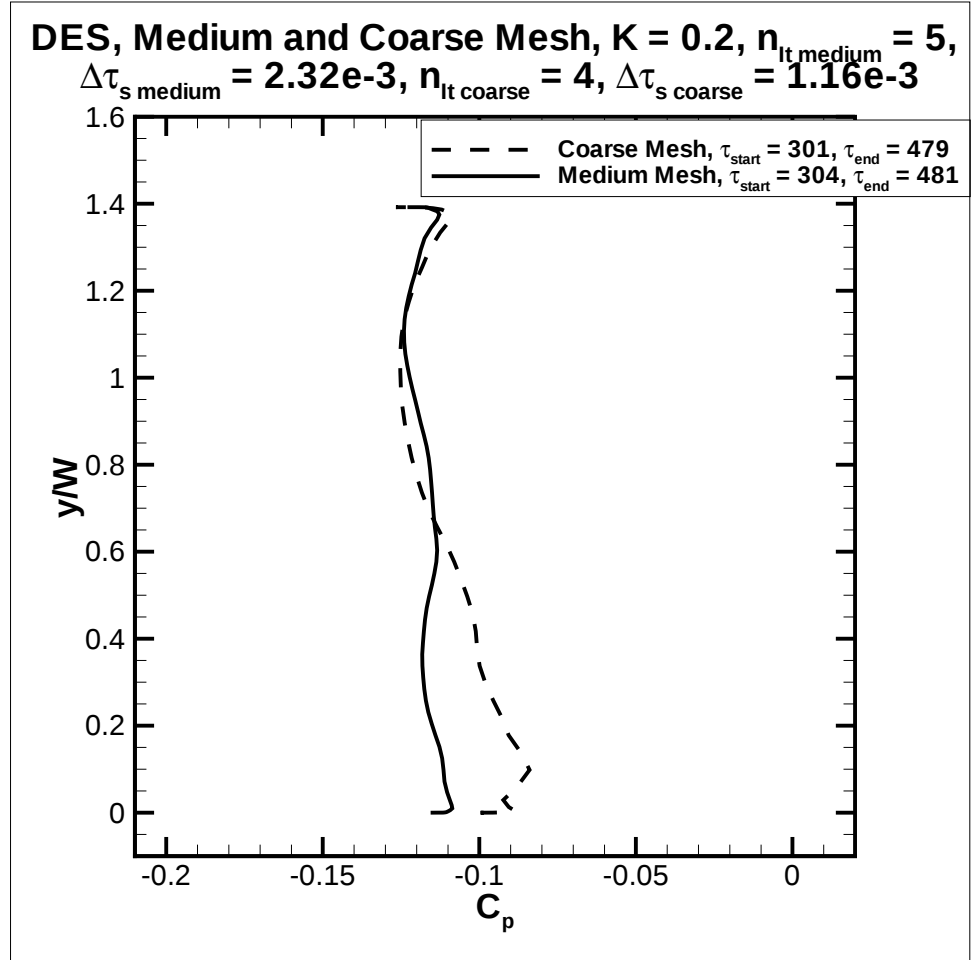


Figure 4.22: Pressure distribution along line T obtained from the coarse and medium mesh simulations

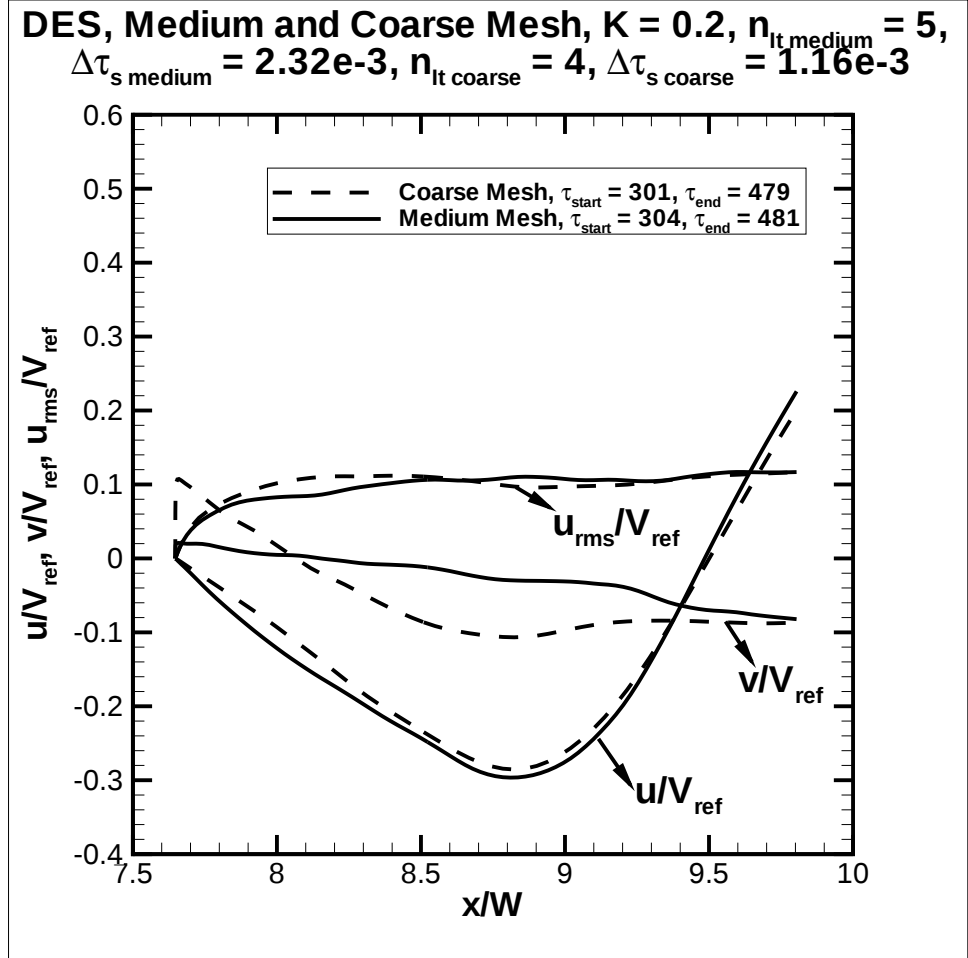


Figure 4.23: Velocity profiles along line A obtained from the coarse and medium mesh simulations

CHAPTER 5

RESULTS AND DISCUSSION

The results from the simulations will be compared with the experimental data in this chapter. The reason for this being that the medium mesh results are closer to the experiments than the coarse mesh results.

5.1 Contour and Streamline Plots

Figure 5.1 shows the various planes in the wake of the GTS model from which the velocity data from the experiments and the simulations are compared. Figures 5.2 to 5.7

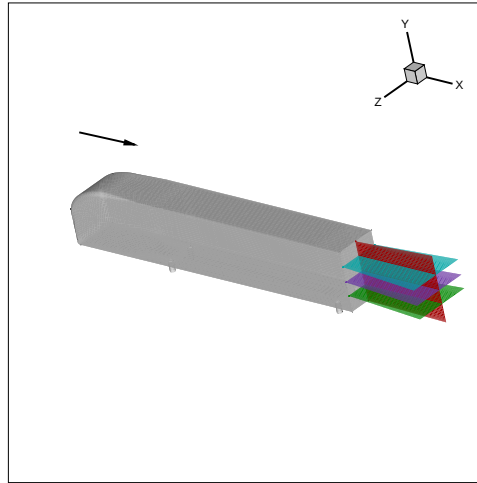


Figure 5.1: Planes used for comparison of velocity data

show a comparison of contour and streamline plots from experiment and medium mesh simulations. The rectangular window shown in the contour plots from the simulations is

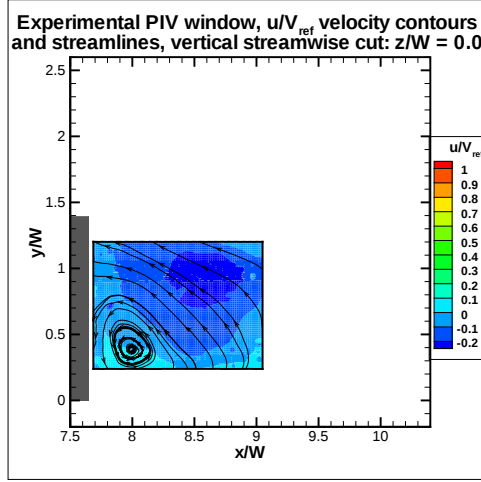


Figure 5.2: Vertical streamwise PIV window in the experiment

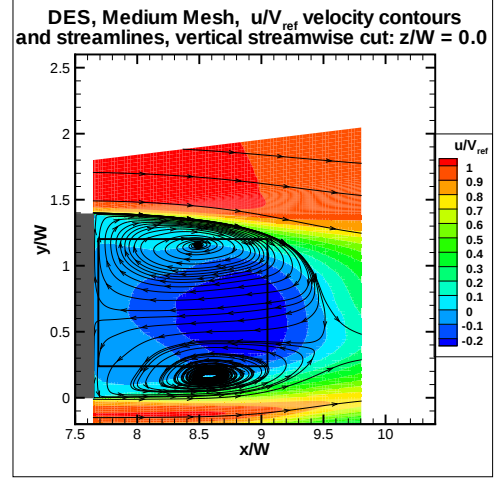


Figure 5.3: Vertical streamwise contour plot from the medium mesh simulations

the window corresponding to the PIV data plane in the experiment. All the data planes shown in the figures are near the base of the GTS model. The gray colored rectangular block seen in all the contour plots is the base of the GTS model. Figure 5.2 shows the vertical streamwise PIV data plane (the red colored plane in the figure 5.1) from the experiment. Figure 5.3 shows the corresponding contour plot from the simulations of the medium mesh. These two figures show the u/V_{ref} velocity contours and streamlines in the vertical streamwise plane at $z/W = 0.0$. The lower vortex is located at $x/W \approx 8.0$ and $y/W \approx 0.38$ in the experiment and at $x/W \approx 8.6$ and $y/W \approx 0.2$ in the simulations. Thus the experimental data shows the lower vortex located much closer to the base of the GTS model and at a higher location than the simulations. The upper vortex is not seen in the experimental PIV window but the nature of the streamlines in the window suggests that it may be located near the upper right corner of the window.

The distribution of the vortices in the medium mesh simulations is much more symmetric as seen in figure 5.3. This difference in location of the lower and the upper vortices from the experiment and the simulations gives a major difference in the pressure distribution along the base of the GTS model.

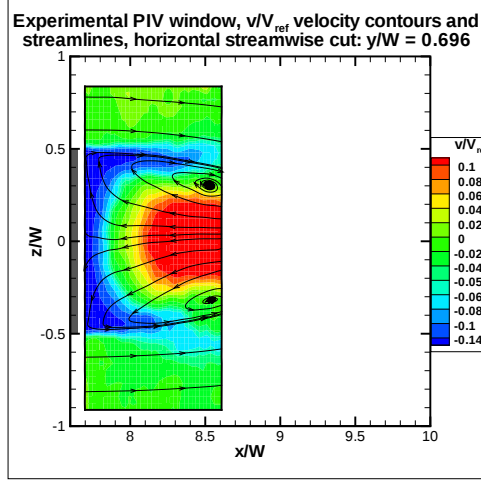


Figure 5.4: Horizontal streamwise PIV window in the experiment ($y/W = 0.696$)

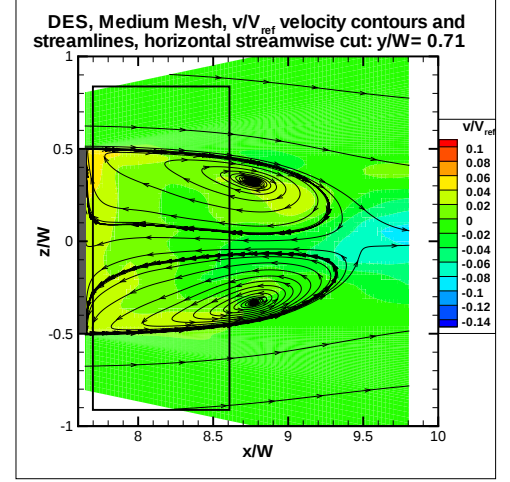


Figure 5.5: Horizontal streamwise contour plot from the medium mesh simulations ($y/W = 0.709$)

Figure 5.4 shows a horizontal streamwise data plane (horizontal violet colored plane in figure 5.1) from the experiment at $y/W = 0.696$ showing the v/V_{ref} velocity contours and streamlines. The corresponding contour plot from the simulations is shown in the figure 5.5. The v/V_{ref} velocity near the base of the GTS model (area near $x/W = 7.7$ and $y/W = 0$) is going down in the experimental PIV window whereas the simulations show an opposite trend. As we move away from the GTS model (area near $x/W = 8.5$ and $z/W = 0$) the PIV window shows the v/V_{ref} velocity coming up but the simulations show an opposite trend. Thus the vertical flow direction from the experiment and the simulations

does not match in this horizontal streamwise plane. This is due to the difference in locations of the upper and lower vortices in the experiment and the simulations as seen in figures 5.2 and 5.3.

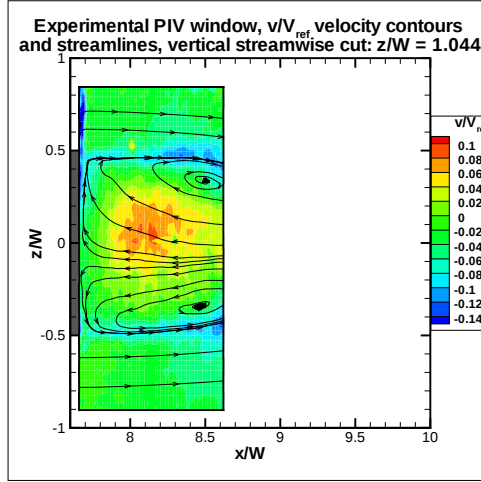


Figure 5.6: Horizontal streamwise PIV window in the experiment ($y/W = 1.044$)

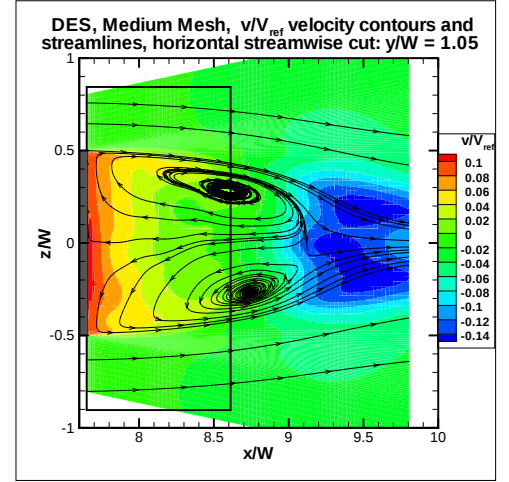


Figure 5.7: Horizontal streamwise contour plot from the medium mesh simulations ($y/W = 1.05$)

Figure 5.6 shows the v/V_{ref} velocity contours and streamlines in a horizontal streamwise plane at $y/W = 1.04$ (horizontal blue colored plane in figure 5.1) from the experiment and figure 5.7 shows the corresponding plot from the simulations. Even in this case there is a mismatch of the v/V_{ref} velocity flow directions near and away from the GTS model when a comparison is made between the experimental plot and the simulations.

5.2 Velocity Profiles

Figures 5.8 to 5.16 show a comparison of the various velocity profiles at different locations in the wake between the experiment and the medium mesh simulations. Uncertainty bars have been placed on the profiles from the simulations. These uncertainty bars take into account the numerical uncertainty due to the statistical convergence, initial transients, effect of sub-iterations (n_{It}) and effect of the time step used for the simulations ($\Delta\tau_s$). Table 5.1 shows the total uncertainty associated with the various flow variables. It should be noted that the banded profiles seen for all the plots from 5.8 to 5.16 show the upper and lower limit due to the numerical uncertainty. Experimental uncertainty is not available for any data. The profiles from the simulations are compared with experimental data available from two PIV windows. One of the data sets is obtained from the vertical PIV window, in the $z/W = 0$ plane (red colored plane in figure 5.1) and the other dataset is obtained from one of the horizontal streamwise PIV windows corresponding to either $y/W = 0.348$, $y/W = 0.696$ or $y/W = 1.044$ (Green, Violet or Blue colored plane in the figure 5.1, respectively). Ideally the data from the two PIV windows should overlap each other, but it can be seen in all the plots from 5.8 to 5.16 that there is a discrepancy in the experimental data used from the two PIV windows. The reason for this discrepancy is not known. Figure 5.8 to 5.10 compare the u/V_{ref} velocity profiles at different locations in the wake. At no location is the u/V_{ref} profile from the simulations matching with either of the experimental datasets. Also

generally the recovery of the u/V_{ref} velocity from the experiments starts earlier than for the simulations.

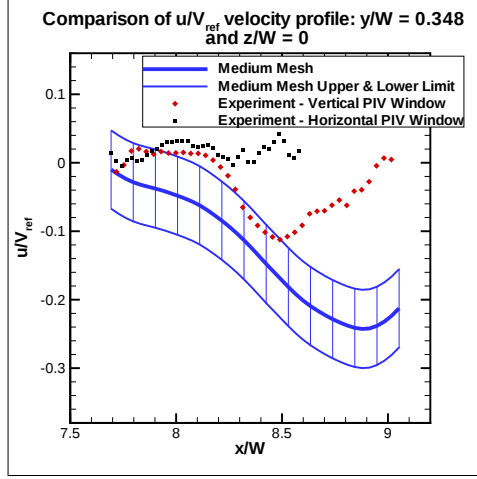


Figure 5.8: Comparison of u/V_{ref} profiles from the experiment and simulations at $y/W = 0.348$ and $z/W = 0$

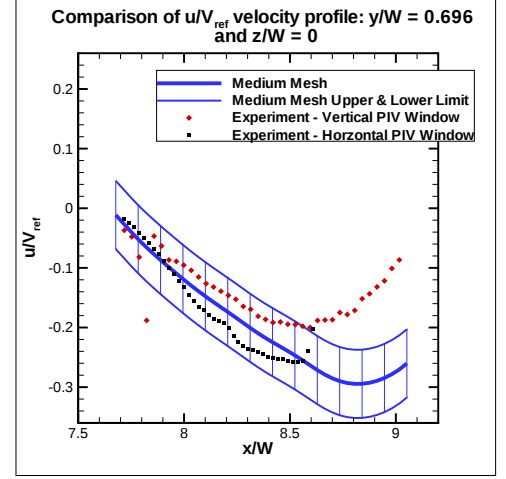


Figure 5.9: Comparison of u/V_{ref} profiles from the experiment and simulations at $y/W = 0.696$ and $z/W = 0$

Figures 5.11 to 5.13 compare the v/V_{ref} velocity profiles from the simulations with the experimental data from the PIV windows. The profile from the simulations shows a completely different trend than found in the data. The main reason of the mismatch of the experimental data with the velocity profiles is the difference in locations of the lower and upper vortices seen in figures 5.2 and 5.3 from the experiment and the simulations respectively.

Figures 5.14 to 5.16 compare the w/V_{ref} velocity profiles from the simulations with the data from the PIV windows. Since these plots are in the $z/W = 0$ plane which is the symmetry plane of the GTS model the cross flow velocity (w/V_{ref}) should be ideally

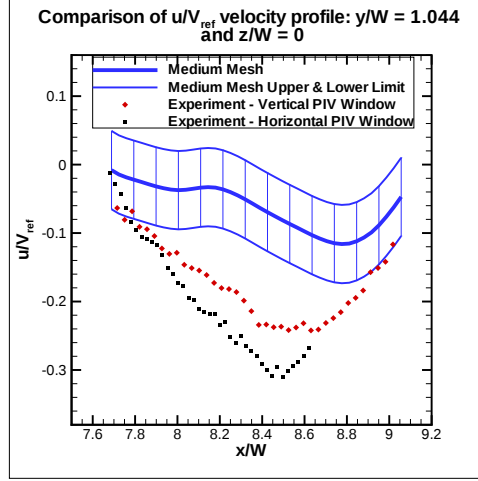


Figure 5.10: Comparison of u/V_{ref} profiles from the experiment and simulations at $y/W = 1.044$ and $z/W = 0$

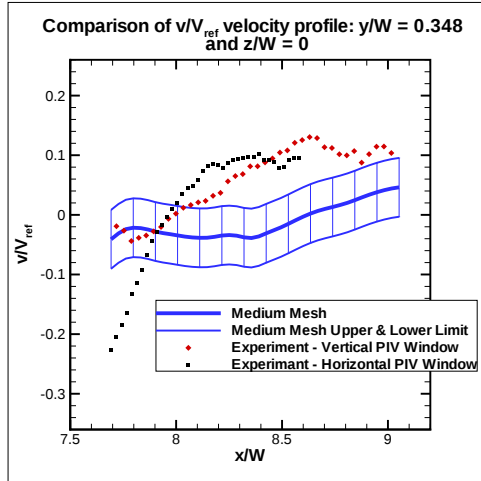


Figure 5.11: Comparison of v/V_{ref} profiles from the experiment and simulations at $y/W = 0.348$ and $z/W = 0$

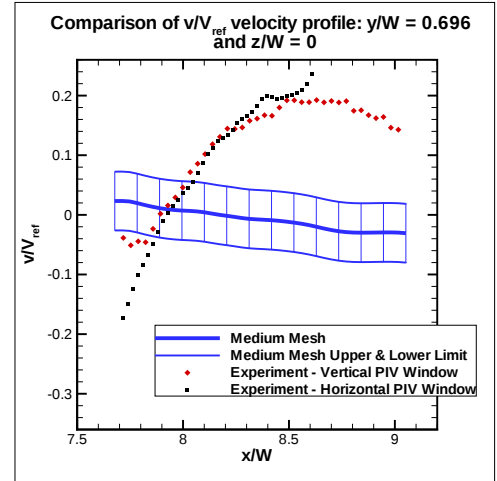


Figure 5.12: Comparison of v/V_{ref} profiles from the experiment and simulations at $y/W = 0.696$ and $z/W = 0$

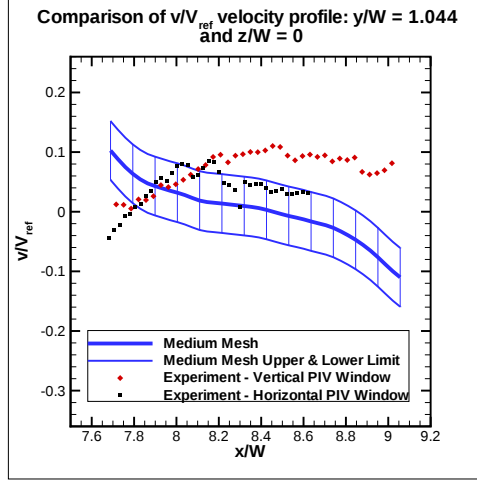


Figure 5.13: Comparison of v/V_{ref} profiles from the experiment and simulations at $y/W = 1.044$ and $z/W = 0$

zero. The experimental data from the PIV windows is generally in the uncertainty limit of the profile from the simulations.

Table 5.1: Total uncertainty values of the velocity and pressure from the simulations

Source of uncertainty	C_p	u/V_{ref}	v/V_{ref}	w/V_{ref}
Initial transients	0.00246	0.01557	0.01524	0.0212
Statistical convergence	0.0047	0.00665	0.01103	0.01396
Number of sub-iterations (n_{It})	0.007	0.01915	0.0150	0.02296
Time step ($\Delta\tau_s$)	0.0175	0.05116	0.0430	0.00564
Total uncertainty	0.02	0.0572	0.0493	0.0346

Table 5.1 shows the various uncertainties in the velocity and pressure profiles from the simulations. These uncertainties are the biggest absolute errors calculated by finding the difference between the current solution and the best available solution. The total uncertainty is obtained by summing the square the various uncertainties and finding the

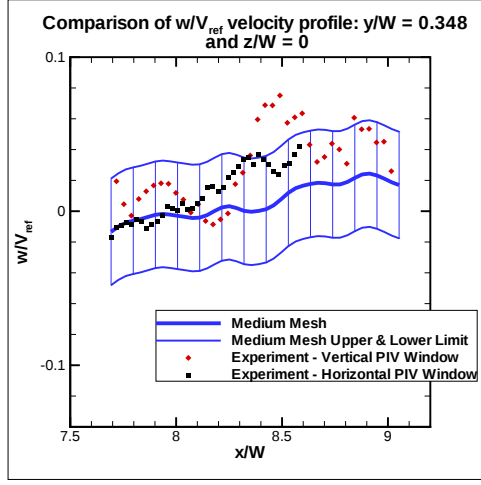


Figure 5.14: Comparison of w/V_{ref} profiles from the experiment and simulations at $y/W = 0.348$ and $z/W = 0$

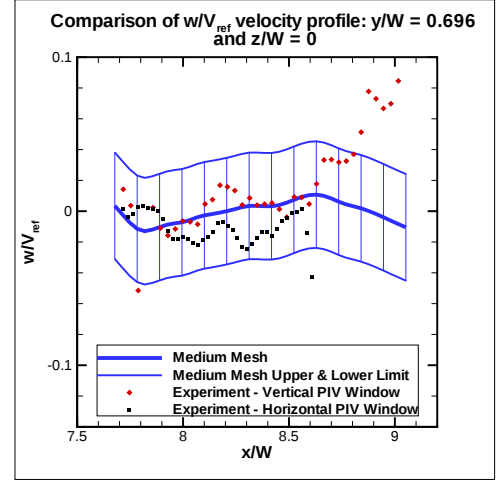


Figure 5.15: Comparison of w/V_{ref} profiles from the experiment and simulations at $y/W = 0.696$ and $z/W = 0$

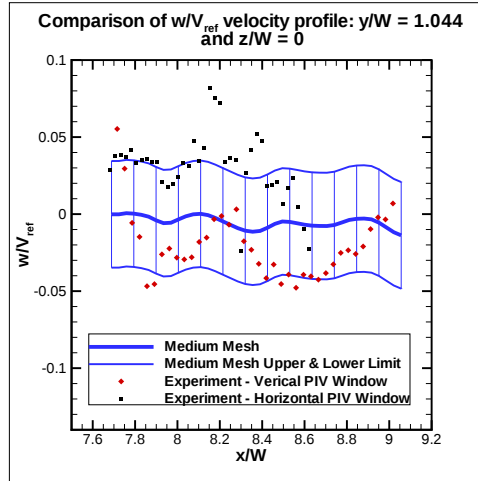


Figure 5.16: Comparison of w/V_{ref} profiles from the experiment and simulations at $y/W = 1.044$ and $z/W = 0$

root of the sum.

$$U_{total} = \sqrt{U_{stat}^2 + U_{tran}^2 + U_{It}^2 + U_{\Delta\tau_s}^2} \quad (5.1)$$

5.3 Pressure Distribution Profile

Figure 5.17 compares the pressure distribution along the base of the GTS model at $z/W = 0.0$. The reason for the discrepancy between the pressure distribution from the experiment and the simulations can again be traced back to the difference in location of the lower and the upper vortices near the base of the GTS model (see figures 5.2 and 5.3) from the experiment and simulations.

In the experiment the lower vortex is closer to the base of the GTS model and hence the experiment shows a large negative pressure in the lower base region. The upper vortex is located much farther from the base of the GTS model hence the pressure in the upper part of the base is higher. The medium mesh simulations on the other hand have the lower and upper vortices almost of equal size and equidistant from the base of the model. Therefore the pressure distribution from the simulations is almost a flat curve having a minor bulge in the negative direction at the locations where the vortices are located. We conclude that the overall magnitude of the C_p predicted by the simulations matches that of the C_p predicted by the experiment but the profiles do not match. It should be noted here that the scale of the C_p profile in the figure 5.17 is enlarged to highlight the differences. Unaune et al. [29] has compared the C_p profile

along the base of the GTS from the simulations and the experiment on a very large scale and hence the plot doesn't show any details of comparison.

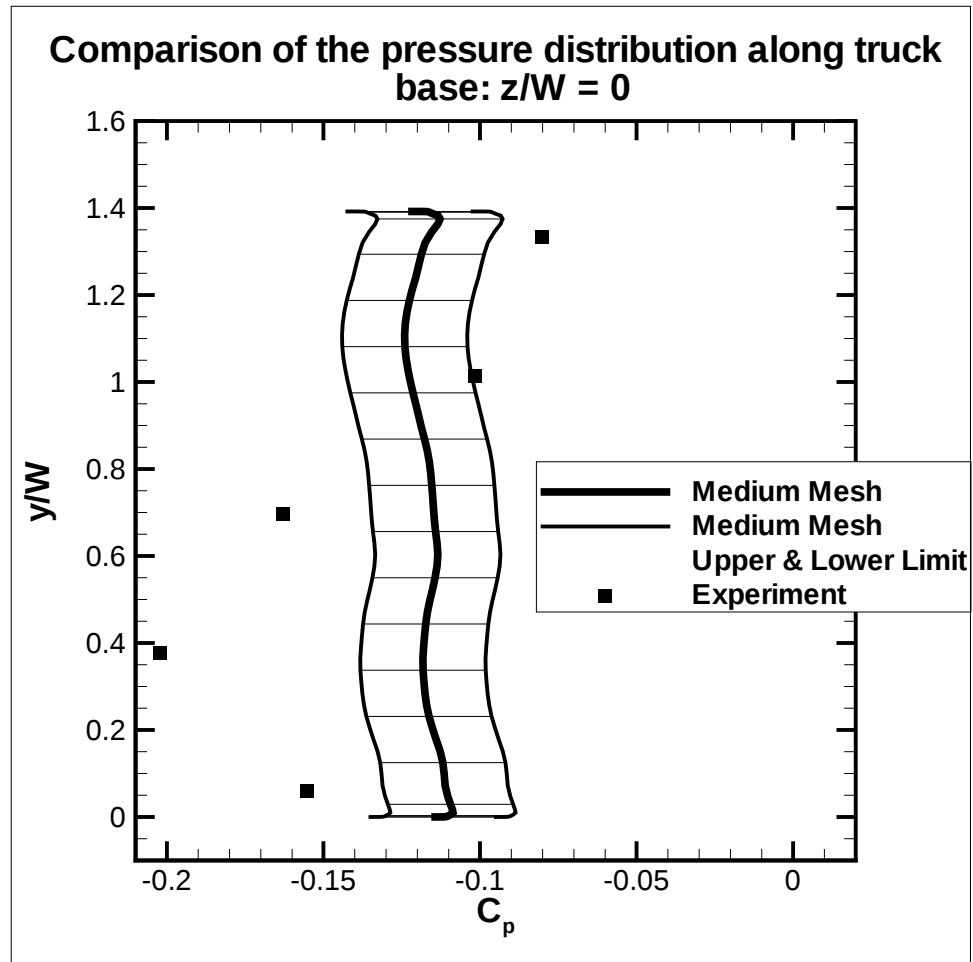


Figure 5.17: Comparison of pressure distribution along base of the truck from the experiment and the medium mesh simulations

Figures 5.18 to 5.21 compare the pressure distribution over the front, top, side and bottom surfaces of the GTS model from the simulations with the experimental data. It

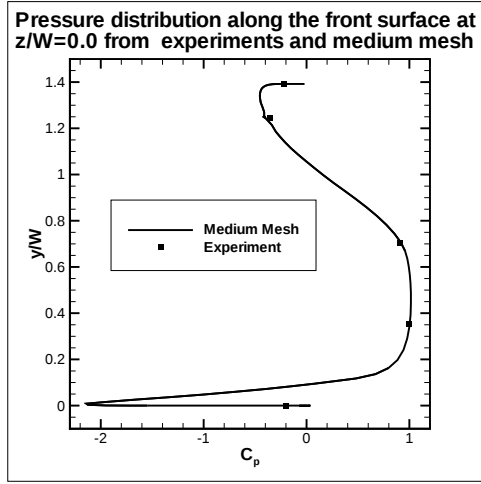


Figure 5.18: Comparison of pressure distribution on the front surface of the GTS model

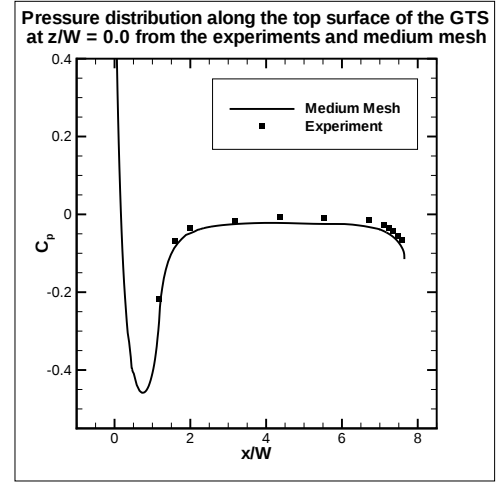


Figure 5.19: Comparison of pressure distribution on the top surface of the GTS model

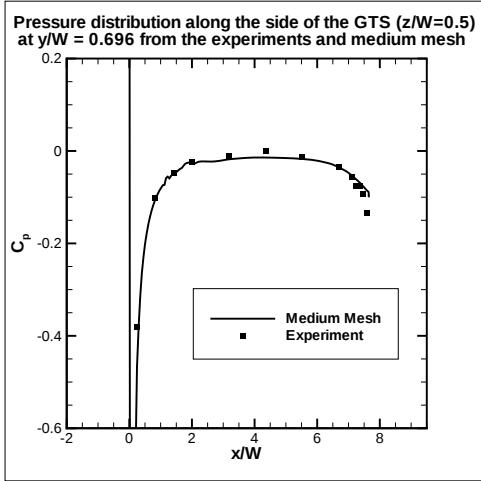


Figure 5.20: Comparison of pressure distribution along the right side of the GTS model

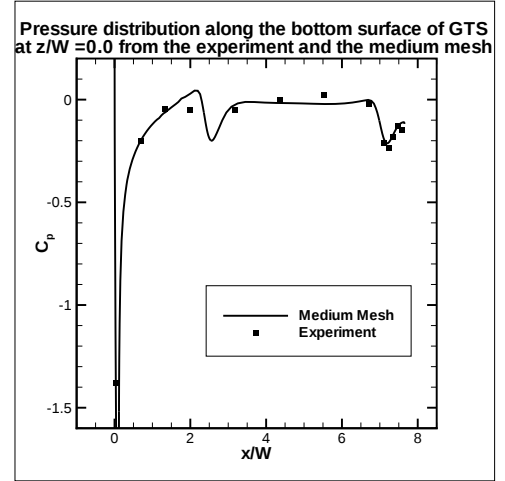


Figure 5.21: Comparison of pressure distribution on the bottom surface of the GTS model

should be noted that the flow over these surfaces is attached, unlike the massively separated near wake flow. The pressure distribution obtained from the simulations matches very well with the experiment in these attached boundary layer regions.

5.4 Drag, Lift and Side Force Coefficients

Table 5.2 compares the drag, side and lift force coefficients predicted by the coarse and medium mesh DES and other drag coefficient values available in the literature with the experimental data. The wind tunnel and the GTS model constructed in the simulations is symmetric about the $z/W = 0.0$ plane so the side force predicted by the coarse and the medium mesh simulations should ideally be zero. The small non-zero values represent statistical error. Figures 5.22 and 5.23 from the coarse and medium meshes respectively, show the variation of the instantaneous and the mean drag with time. Note the higher amplitude oscillations of the medium mesh compared to the coarse mesh. This may be due to the better spatial resolution provided by the medium mesh.

Table 5.2: Comparison of the drag, side and lift force coefficients

	(C_{Dmean})	(C_{Smean})	(C_{Lmean})
Experiment	0.249±0.01	0.007	-0.158
Coarse Mesh DES	0.2565	-1e-4	-0.1187
Medium Mesh DES	0.2439	-3e-5	-0.1224
Unaune et al. [29] DES	0.253	NA	NA
Maddox et al. [18] DES	0.279	NA	NA
Roy et al. [21] (RANS Menter $k - \omega$)	0.298	NA	NA
Roy et al. [21] (RANS Spalart-Allmaras)	0.413	NA	NA

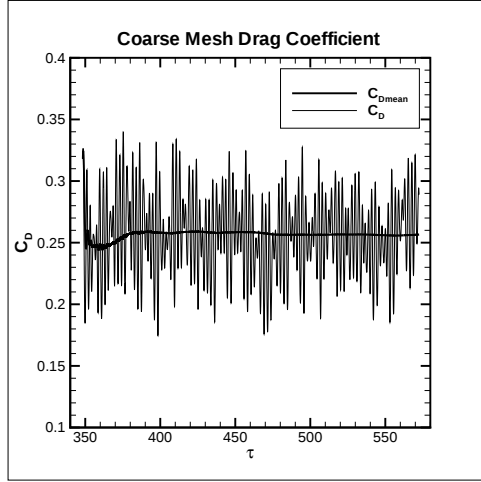


Figure 5.22: Coarse mesh mean and instantaneous drag coefficient

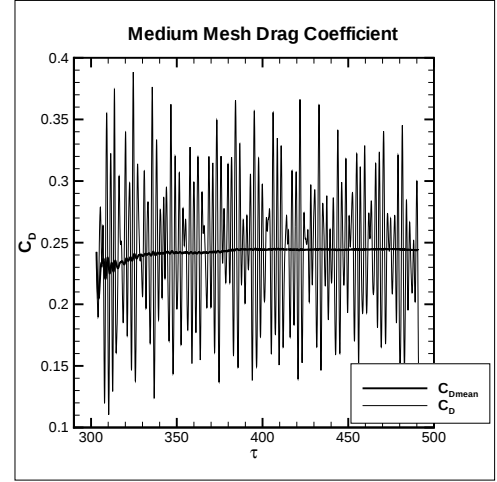


Figure 5.23: Medium mesh mean and instantaneous drag coefficient

5.5 Vortex Core Locations

Figures 5.24, 5.24 and 5.26 compare the location of the vortex cores in vertical and horizontal streamwise plane between the experiment and the simulations. The location of the upper vortex cannot be seen in the PIV window in the vertical streamwise plane (red colored plane in figure 2.5) from the experiment but it may be located in the upper right corner outside the PIV window shown in figure 5.2. While the prediction of the location of the vortex cores improves from the coarse mesh to the medium mesh in the vertical streamwise plane ($z/W = 0$), the same cannot be said about the horizontal streamwise planes. However it should be noted that the vortex cores in the horizontal planes from the medium and coarse mesh are already very close to the experiment as compared to what we see in the vertical streamwise plane.

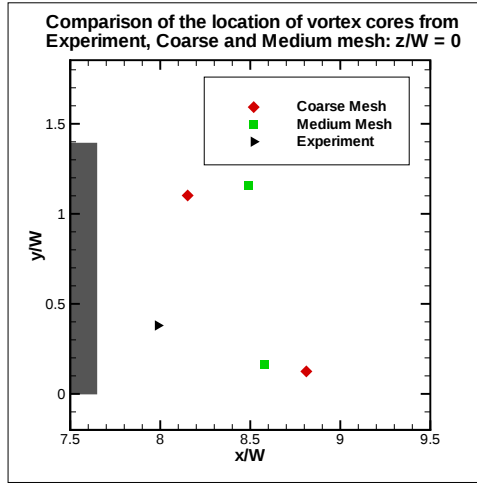


Figure 5.24: Location of vortex cores in vertical streamwise plane ($z/W = 0.0$)

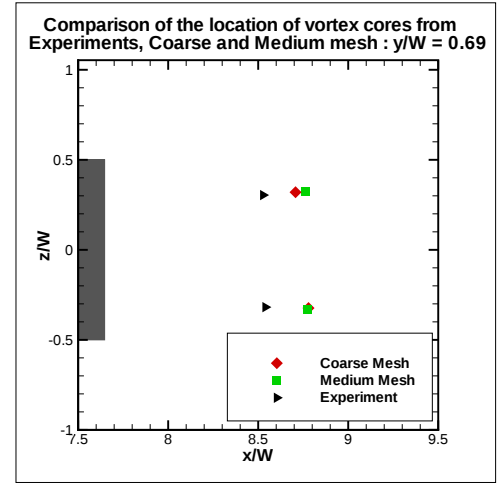


Figure 5.25: Location of vortex cores in horizontal streamwise plane ($y/W = 0.696$)

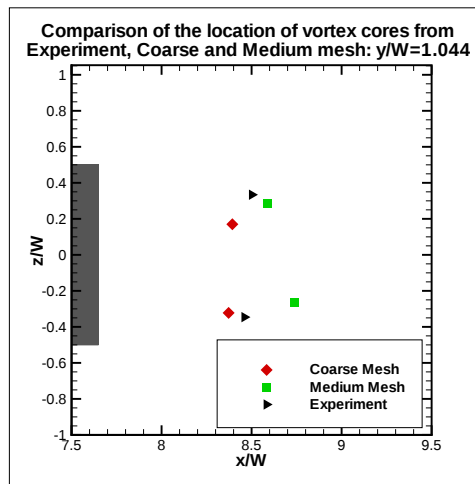


Figure 5.26: Location of vortex cores in horizontal streamwise plane ($y/W = 1.044$)

5.6 Turbulent Eddies in the Wake

Figure 5.27 shows the instantaneous z-vorticity in the vertical streamwise plane (top) and the isosurfaces of z-vorticity in the wake of the GTS model (bottom) for the medium grid. The z-vorticity is a measure of the rotation of the flow about the z-axis,

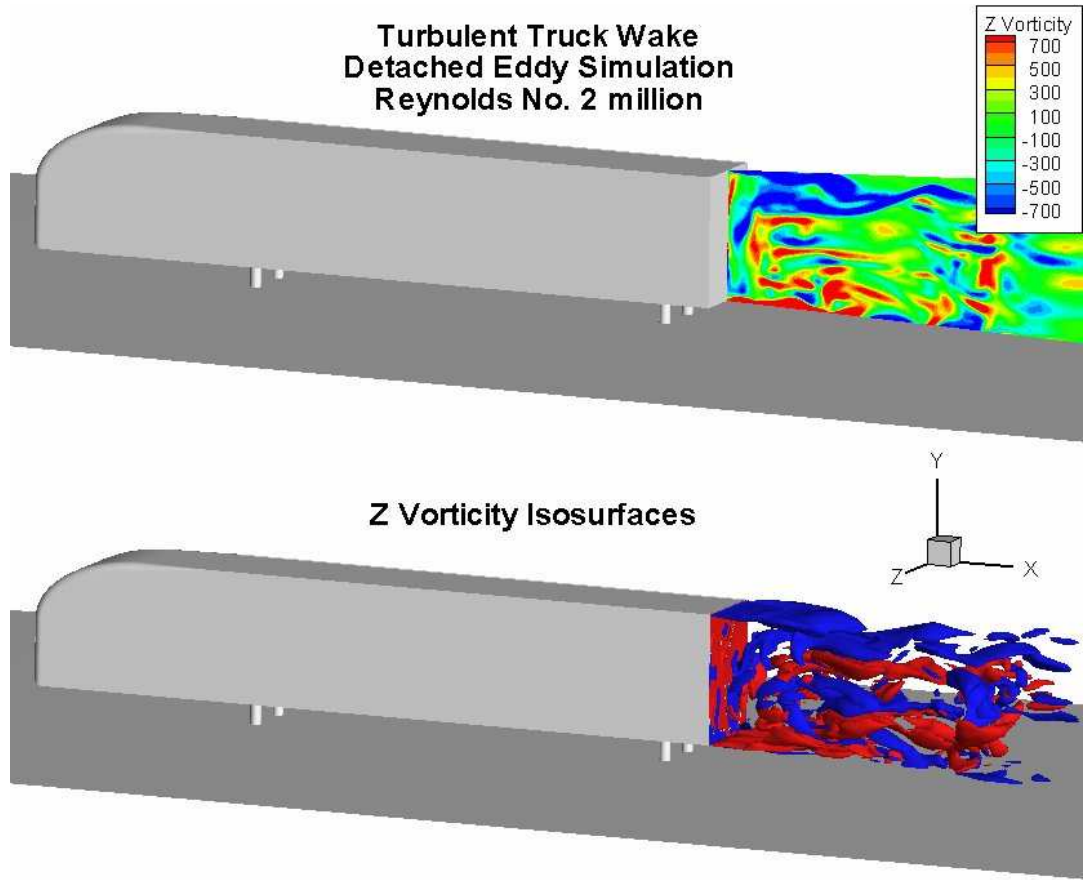


Figure 5.27: Vortical structures resolved using LES in the wake of the GTS on the medium mesh

red counter-clockwise rotation and blue denotes the clockwise rotation. Figure 5.27 gives

an indication of the size and nature of the turbulent structures in the wake resolved by the DES model on the medium mesh (13.2 million grid cells).

5.7 Power Spectra of the Unsteady Pressure Signal

In the experiment there was one high-frequency pressure transducer located on the base of the GTS model ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$). The power spectra for the unsteady pressure signal is obtained by performing a fast Fourier transformation (FFT) of the normalized pressure signal from the experiment and the simulations. FFT is an efficient way to perform a discrete Fourier transform of a signal which converts the signal from the time domain into the frequency domain. The power spectra serves the purpose of giving an idea about the frequency of the turbulent structures in the region near the base of the GTS model. The frequency corresponding to the peak in the power spectra is the frequency containing the most turbulent energy. FFT of the pressure signals at other locations in horizontal streamwise plane ($y/W = 0.696$) and vertical streamwise plane ($z/W = 0$) from the coarse mesh simulations is also performed. A comparison of the power spectra from the experiment and from the simulations will give an idea about the frequency of the turbulent eddies from the experiment and the simulations. Figures 5.28, 5.29 and 5.30 show the power spectra obtained from the experiment and the corresponding spectra from coarse mesh and medium mesh simulations, respectively. The y axis in the plots denotes the energy associated with the unsteady

turbulent wake in units of decibel (dB). The power spectra obtained from the experiment shows a peak frequency at nearly 24Hz. The red line shown in all three plots is the best fit curve of the FFT of the experimental data. It shows the general trend of the FFT of the experimental data. The FFT plots from the simulations (Figures 5.29 and 5.30) show three peaks at approximately 80 Hz, 110 Hz and 130 Hz. Also some peaks are seen in the range of 10 to 30 Hz which are highlighted by the circle and appear to roughly correspond to the dominant frequencies found in the experiment. Figures 5.31 to 5.34 show power spectra of the pressure signals at four locations in the wake from the coarse mesh simulations. These locations at $x/W \approx 8.15$ (approximately $W/2$ away from the trailer base) in the regions where the attached boundary layers on the side, top and bottom surfaces of the GTS model separate from the trailer base. Figures 5.31 and 5.32 show the spectra obtained from the pressure signals in the separated side-wall shear layers; however, there are no distinct peaks seen in these two plots. Figures 5.33 and 5.34 give the power spectra in the shear layers which separate from the bottom and top of the truck, respectively and show distinct peaks which roughly correspond to the peaks seen in the figures 5.29 and 5.30. Animations of the turbulent flow in the wake were performed and they showed the mixing of the shear layers from the separated boundary layers on the side, top and bottom surfaces of the GTS with the LES region in the wake. The peaks seen in figure 5.33 and 5.34 appear to correspond to the large-scale unsteadiness of the shear layers seen in the animations. The peaks seen in the FFT plots from the simulations thus seem to correspond to the dominant shedding frequency of the shear

layers generated by the boundary layers on the top, side and bottom surfaces of the GTS model. This problem does not appear in the experimental data and highlights one of basic drawbacks of the DES model, namely that there is no mechanism for converting the modeled turbulence from the attached RANS boundary layers into resolved fluctuations in the LES wake region. Strouhal number based on the width of the trailer was also calculated. The FFT performed on the normalized drag plot gave a peak frequency of ≈ 132 Hz. Since the FFT plots of the pressure signal also showed peak frequency of ≈ 132 Hz this frequency is used in the calculation of Strouhal number. The Strouhal number from the computations is ≈ 0.45 . The Strouhal number from the experimental data is 0.08.

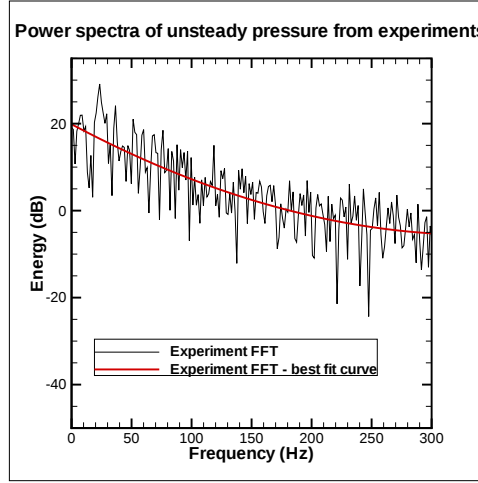


Figure 5.28: Fast Fourier transform of the pressure signal from experimental data ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$)

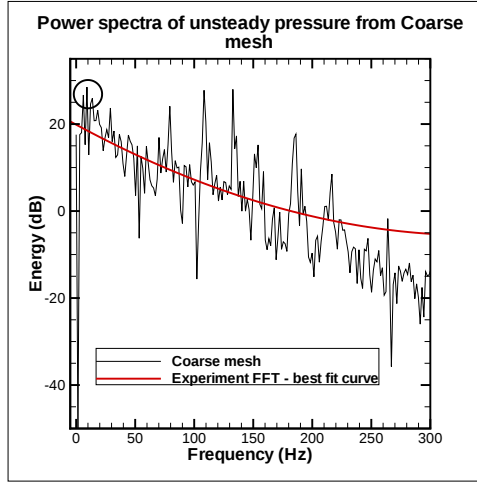


Figure 5.29: Fast Fourier transform of the pressure signal from coarse mesh simulations ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$)

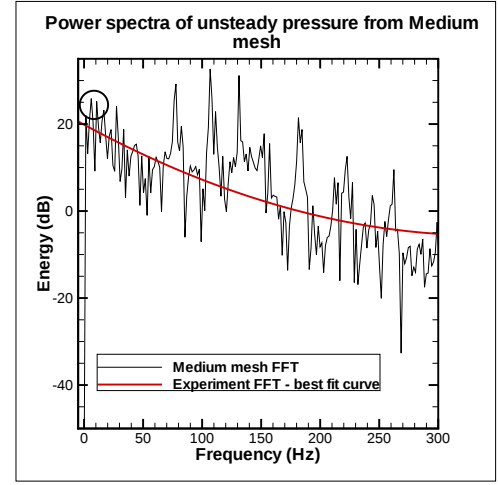


Figure 5.30: Fast Fourier transform of the pressure signal from medium mesh simulations ($x/W = 7.647$, $y/W = 0.63$ and $z/W = -0.46$)

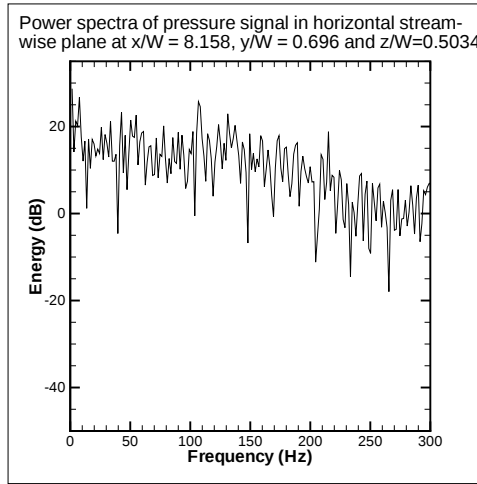


Figure 5.31: Fast Fourier transform of the pressure signal in the horizontal streamwise plane from the coarse mesh simulations

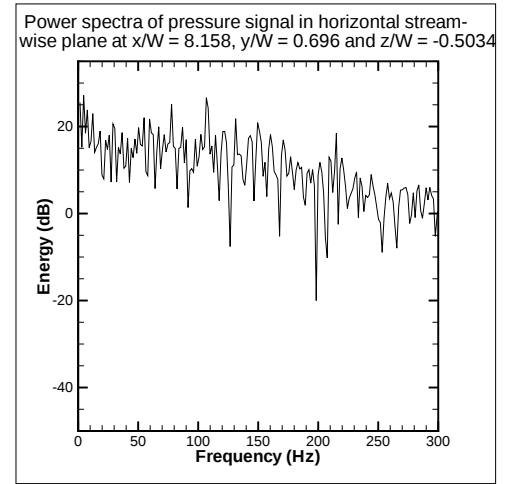


Figure 5.32: Fast Fourier transform of the pressure signal in the horizontal streamwise plane from the coarse mesh simulations

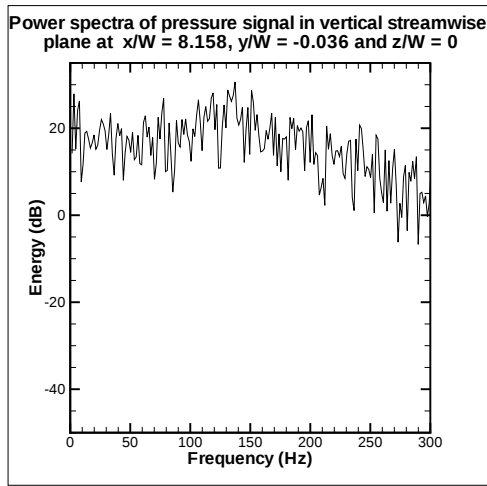


Figure 5.33: Fast Fourier transform of the pressure signal from the vertical streamwise plane in the coarse mesh and near the bottom surface of the GTS model

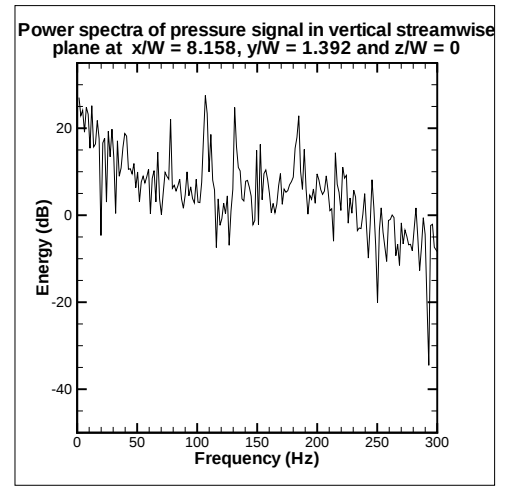


Figure 5.34: Fast Fourier transform of the pressure signal from the vertical streamwise plane in the coarse mesh and near the top surface of the GTS model

CHAPTER 6

CONCLUSION

The initial transient period of $\tau \approx 300$ ($t \approx 1$ s) was quite long, thus making the simulations computationally expensive than expected. The time window required for achieving statistically converged results was found to be $\Delta\tau_w \approx 180$ ($\Delta t_w \approx 0.6$ s). These estimates for the initial transient period and time window required for statistical convergence will help in future DES simulations. Based on this finding we concluded that similar studies made by Unaune et al. [29] and Maddox et al. [18] have not marched long enough in the physical time during their computations to get out of this initial transient period and neither have they collected the statistics for a long enough period to get statistically converged results. Analysis of the number of sub-iterations (n_{It}) required to achieve an iteratively converged solution suggested that the results from the case with 4 sub-iterations are reasonably close to the solution obtained from 8 sub-iterations; however computations performed with 1 sub-iteration did not give good results. It is concluded that 4 sub-iterations are sufficient for achieving accurate results when a non-dimensional time step of $\Delta\tau_s = 1.16 \times 10^{-3}$ ($\Delta t_s = 4 \times 10^{-6}$ s) is used. It should be noted that this analysis was performed using the coarse mesh simulations. It is computationally expensive to use the medium mesh for this analysis, so we have assumed that the conclusions drawn for the required number of sub-iterations hold true for the medium mesh results as well. Analysis of the the time step ($\Delta\tau_s$) that should be

used for the simulations suggested that the results from the $\Delta\tau_s = 2.32 \times 10^{-3}$ ($\Delta t_s = 8 \times 10^{-6}$ s) case were significantly different than the results from the $\Delta\tau_s = 1.16 \times 10^{-3}$ ($\Delta t_s = 4 \times 10^{-6}$ s) case. Thus a maximum time-step of $\Delta\tau_s = 1.16 \times 10^{-3}$ should be used instead of $\Delta\tau_s = 2.32 \times 10^{-3}$ to get accurate results. The grid size has a significant effect on the results from the simulations. The medium mesh results are much better than the coarse mesh simulations.

The drag value predicted by the simulations matches very well with the experimental value. The pressure values predicted by the DES in the attached boundary layer regions also match very well with the experiment. The range of the pressure values predicted by the DES approach in the base of the GTS model matches with the range of the values of pressure from the experiments. The actual pressure profile, however, does not match with the experimental data. The prediction of the location of the vortices near the base of the model does not match with the experimental data. Here the DES model fails to accurately capture the details of the turbulent flow in the near base region. The inability of these DES simulations to accurately predict the details of the turbulent wake structure suggests that this model may not be able to accurately predict the drag when drag reduction devices are added to the base. The failure of the current DES simulations to predict wake details appear to be related to one of the main drawbacks of the hybrid RANS/LES methods, namely that there is no mechanism to transfer information from the RANS region (i.e., the boundary layer) to the LES region (i.e., the wake). Hence when the shear layers in the trailer base enter the wake region governed by the LES

model, there is no way to convert the large, modeled eddy viscosity into large-scale turbulent fluctuations required for the LES model. Animations of these RANS-based shear layers confirmed that the large turbulent energy peaks seen in the 80-150 Hz range correlate with the frequency of the shear layer flapping instability. The experimental data did not show these features. The DES model is cheaper than the LES approach and the Direct Numerical Simulations, but is still computationally expensive.

CHAPTER 7

RECOMMENDATIONS

It is expected that the fine mesh simulations will give better comparison with experiments, hence fine mesh simulations should be performed. Research on the transfer of information from the modeled turbulence obtained from RANS equations in the boundary layer to the LES model in the separated region should improve the predictions of the wake flow details using DES on coarser grids. The front portion of the coarse mesh should be truncated and the results from the truncated coarse mesh should be compared with the results from the full geometry coarse mesh. If these simulations show good agreement then it can be concluded that the front portion of the GTS does not play a significant role in the turbulent flow in the near-base region. Then the front portion of the GTS can be truncated to make the fine grid simulations computationally less expensive. The effect of drag reduction devices such as the boattail plates which are attached to the base of the model can then be studied using DES on the truncated mesh. Finally, truncating the front portion of the GTS mesh will allow the attached RANS boundary layers to be perturbed so they contain the proper large-scale turbulent fluctuations when they reach the LES region.

BIBLIOGRAPHY

- [1] MacCallen, R., Salari, K., Ortega, J., Browand, F., Hammache, M., Hsu, T., Leonard, A., Chatelain, P., Rubel, M., Roy, C. J., DeChant, L., Hassan, B., Heineck, J. T., Yaste, D., Storms, B., Englar, R., and Pointer, D., 2004, "DOE's Effort to Reduce Truck Aerodynamic Drag - Joint Experiments and Computations Lead to Smart Design," *34th AIAA Fluid Dynamics Conference*, June 28 - July 1. Portland, OR.
- [2] Cooper, K. R., 2003, "Truck Aerodynamics Reborn - Lessons from the Past," SAE Technical Paper Series 2003-01-3376.
- [3] US DOE Transportation Energy Data Book: Edition 23, 2003.
- [4] Storms, B. L., James, J. C., Heineck, J. T., Walker, S. M., Driver, D. M., and Zilliac, G. G., 2001, "An Experimental Study of the Ground Transportation System (GTS) Model in the Nasa Ames 7 - by 10 - ft Wind Tunnel," NASA/TM-2001-209621.
- [5] Storms, B. L., Satran, D. R., Heineck, J. T., and Walker, S. M., 2004, "A Study of Reynolds Number Effects and Drag-Reduction Concepts on a Generic Tractor-Trailer," AIAA 2004-2251.
- [6] Wilcox, D. C., "Turbulence Modeling for CFD" DCW Industries Inc, 5354 Palm Drive, La Cañada, CA, November 1994, pp. 5-19, 322-327.
- [7] Camarri, S., Salvetti, M. V., Koobus, B., and Dervieux, A., 2004 "A hybrid RANS/LES approach applied to Bluff-body flow simulation," *European Congress on Computational Methods in Applied Sciences and Engineering*, Jyvaskyla, Finland, July 24-28.
- [8] Spalart, P. R., Jou, W. H., Strelets, M., and Allmaras, S., 1997, "Comments on the Feasibility of LES for Wings and on a Hybrid RANS/LES Approach," *Advances in DNS/LES*, Columbus, OH, Greyden Press.
- [9] Vio, G., Watkins, S., Mousley, P., Watmuff, P., and Prasad, S., 2005 "Flow Structures in the near-wake of the Ahmed model," *Journal of Fluids and Structures*, Vol. 20, pp. 673-695.
- [10] Martinuzzi, R. J., Bailey, S. C., and Knopp, G. A., 2003 "Influence of Wall Proximity on Vortex Shedding from a Square Cylinder," *Experiments in Fluids*, Vol. 34, pp. 585-596.

- [11] Martinuzzi, R. J., and Tropea, C., 1993 “The Flow Around Surface-Mounted, Prismatic Obstacles Placed in a Fully Developed Channel Flow,” *Journal of Fluids Engineering*, Vol. 115, pp. 85-92.
- [12] Lyn, D. A., Einav, S., Rodi, W., and Park, J., 1995 “A Laser-Doppler Velocimetry Study of Ensemble-averaged Characteristics of the Turbulent near Wake of a Square Cylinder,” *Journal of Fluids Mechanics*, Vol. 304, pp. 285-319.
- [13] Ramesh, V., Vengadesan, S., and Narasimhan J. L., 2005 “3D Unsteady RANS Simulations of Turbulent Flow Over Bluff Body by Non-linear Model,” *International Journal of Numerical Methods for Heat & FluidFlow*, Vol. 16, No. 6, pp. 660-673.
- [14] Rodi, W., 1997 “Status of Large Eddy Simulation: Results of a Workshop,” *Journal of Fluids Engineering*, Vol. 119, pp. 248-262.
- [15] Nakayama, A., and Vengadesan, S. N., 2002 “On the Influence of Numerical Schemes and Sub-grid Stress Models on Large-eddy Simulation of Turbulent Flow Past Square Cylinder,” *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 38, No. 3, pp. 227-253.
- [16] Spalart, P. R., and Squires, K. D., 2004 “The Status of Detached-Eddy Simulation for Bluff Bodies,” *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*, R. McCallen, F. Browand and J. Ross, eds., Springer, 19.
- [17] Nikitin, N. V., Nicoud, F., Wasistho, B., Squires K. D., and Squires, K. D., 2000 “An Approach To Wall Modeling in Large Eddy Simulations,” *Physics of Fluids*, Vol. 12, pp. 7-10.
- [18] Maddox, S., Squires, K. D., Wurtzler, K. E., and Forsythe, J. R., 2004 “Detached-Eddy Simulation of the Ground Transportation System,” *The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*, R. McCallen, F. Browand and J. Ross, eds., Springer, 19.
- [19] Squires, K. D., Forsythe, J. R., and Spalart, P. R., 2005 “Detached-Eddy Simulation of the Separated Flow Over a Rounded-Corner Square,” *Journal of Fluids Engineering*, Vol. 127, pp. 959-966.
- [20] Roy, C. J., DeChant, L. J., Payne, J. L., and Blottner, F. G., 2003 “Bluff-Body Flow simulations Using Hybrid RANS/LES,” AIAA 2003-3889.
- [21] Roy, C. J., Payne, J. L., and Payne, M. M, 2006 “RANS Simulations of a Simplified Tractor/Trailer Geometry,” *Journal of Fluids Engineering*, Vol. 128, pp. 1083-1089.

- [22] Lübcke, H., Schmidt, St., Rung, T., and Thiele, F., and 2001 “Comparison of LES and RANS in Bluff-body Flows,” *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 89, pp. 1471-1485.
- [23] Krajnović, S., and Davidson, L., 2002 ,“Large-Eddy Simulation of the Flow Around a Bluff Body,” *AIAA Journal*, Vol. 40, No. 5 pp. 927-935.
- [24] Davidson, L., 1997 ,“Large-Eddy Simulation: A Dynamic One-Equation Sub-grid Model for Three-Dimensional Recirculating Flow,” 11th *International Symposium on Turbulent Flows in Complex Geometry*, Vol. 3, pp. 26.1-26.6.
- [25] Menon, S., Kim, W.-W., 1996, “High Reynolds Number Flow Simulations Using the Localized Dynamic One-Equation Subgrid Model,” AIAA 96-0425.
- [26] Sohankar, A., Davidson, L., and Norberg, C., 2000,“Large Eddy Simulation of Flow Past a Square Cylinder: Comparison of Different Subgrid Scale Models,” *Journal of Fluids Engineering*, Vol. 122, pp. 39-47.
- [27] Nichols, R. H., 2006 ,“Comparison of Hybrid Turbulence Models for a Circular Cylinder and a Cavity,” *AIAA Journal* Vol. 44, No. 6. pp. 1207-1219.
- [28] Barone, M. F., Roy, C. J., 2005 ,“Evaluation of Detached Eddy Simulation for Turbulent Wake Applications,” AIAA Paper 2005-0504.
- [29] Unaune, S. V., Sovani, S. D., and Kim, S., 2005 ,“Aerodynamics of a Generic Ground Transportation System: Detached Eddy Simulation,”*The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains*, R. McCallen, F. Browand and J. Ross, eds., Springer, 19.
- [30] Baysal, O., and Bayraktar, I., 2000 ,“Computational Simulations for the External Aerodynamics of Heavy Trucks,” SAE Technical Paper Series 2000-01-3501.
- [31] Rodi, W., 2000 “Comparison of LES and RANS Calculations of the Flow Around Bluff Bodies,” *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 69-71, pp. 55-75.
- [32] Yee, H. C., Sandham, N. D., and Djomehri, M. J., 1998 “Low-dissipative High-Order Shock-Capturing Methods Using Characteristic-Based Filters,” *Journal of Computational Physics*, Vol. 150, pp. 199-238.
- [33] Harten, A., 1984 “On Class of High Resolution Total-Variation-Stable Schemes,” *SIAM Journal of Numerical Analysis*, Vol. 21, No. 1, pp. 1-23.
- [34] Kato, M., and Launder, B., 1993 ”The modeling of Turbulent Flow around Stationary and Vibrating Square Cylinders,” 9th *International Symposium on Turbulent Shear Flows*, Kyoto.

- [35] Norris, H., and Reynolds, W., 1975 "Turbulent Channel Flow with a Moving Wavy Boundary," Department of Mechanical Engineering. Report FM/10, Stanford University.
- [36] Wong, C. C., Blottner, F. G., Payne, J. L., and Soetrisno, M., 1995 "Implementation of a Parallel Algorithm for Thermo-Chemical Non equilibrium Flow Solutions," AIAA Paper 95-0152.
- [37] INCA User's Manual, Version 2.0, 1995, Amtec Engineering, Inc., Bellevue, WA.
- [38] Payne, J. L., and Walker, M. A., 1995 "Verification of Computational Aerodynamic Predictions for Complex Hypersonic Vehicles using the INCA Code," AIAA Paper 95-0762.
- [39] Roy, C. J., Gallis, M. A., Bartel, T. J., and Payne, J. L., 2003 "Navier-Stokes and DSMC predictions for Laminar Hypersonic Shock-Induced Separations," *AIAA Journal* Vol. 41, No. 6. pp. 1055-1063.
- [40] Germano, M., Piomelli, U., Moin, P., and Cabot, W., 1991 "A Dynamic Sub-grid scale Eddy Viscosity Model," *Physics of Fluids* Vol. 3, No. 7. pp. 1760-1765.
- [41] Smagorinsky, J., 1963 "General Circulation Experiments with the primitive Equations, Part I: The Basic Experiment," *Monthly Weather Review* Vol. 91, pp. 99-165
- [42] Ghosal, S., Lund, T., Moin, P., and Akselvoll, K., 1995 "A Dynamic Localization Model for Large-Eddy Simulation of Turbulent Flows," *Journal of Fluid Mechanics*, Vol. 286, pp. 229-225
- [43] Spalart, P. R., and Allmaras, P. R., 1994 "A One-Equation Turbulence Model for Aerodynamic Flows," *La Recherche Aerospatiale* Vol. 1, pp. 5-21.
- [44] Yoon, S., and Jameson, A., 1987 "An LU-SSOR Scheme for the Euler and Navier-Stokes Equations," AIAA Paper 87-0600.
- [45] Chung, T. J., "Computational Fluid Dynamics" 2002 Cambridge University Press.
- [46] Peery, K. M., and Imlay, S. T., 1986 "An Efficient Implicit Method for Solving Viscous Multi-Stream Nozzle/Afterbody Flow Fields," AIAA Paper 86-1380.
- [47] Roy, C. J., and Blottner, F. G., 2003 "Methodology for Turbulence Model Validation: Application to Hypersonic Flows," *Journal of Spacecrafts and Rockets* Vol. 40, No. 3. pp. 313-325.
- [48] Shih, T. H., and Lumley, J. L., 1997 "A New Reynolds Stress Algebraic Equation Model," *Computational Methods in Applied Mechanical Engineering* Vol. 125, No. 3. pp. 287-302.
- [49] <http://www.cfdreview.com/article.pl?sid=01/04/28/2131215>

[50] http://en.wikipedia.org/wiki/Fast_Fourier_transform

APPENDIX

APPENDIX A

FORTRAN CODE FOR CREATING ANIMATIONS USING TECPLOT

Following is a Fortran code used during creating animations using Tecplot

```
Program DES-Coarse-Mesh-OPLTF-Animat
implicit double precision(a-h,o-z)
OPEN(UNIT=13,FILE='OPLTF11',STATUS='unknown')
OPEN(UNIT=14,FILE='Reduced-OPLTF11',STATUS='unknown')
write(14,*)'TITLE = "INCA Version 2 Input File Created by Gridge "'
write(14,*)'VARIABLES = "X(M)" "Y(M)" "Z(M)" "U(M/S)" "V(M/S)"
& "W(M/S)" "P(N/M2)" "T(K)" "R(KG/M3)" "Mach" "TVisc"'
i1=1
do itstep=1,5
read (13,*)
write(*,*)itstep
read(13,*)
read(13,*)
i1=i1+3
do nzone=1,46
read(13,11)zone,ii,jj
i1=i1+1
if ((mod(ii,5)).eq.0)then
ians1=0
ians2=ii/5
else
ians1=mod(ii,5)
ians2=(ii-ians1)/5
end if
read(13,*)
read(13,*)
i1=i1+2
do nvar=1,11
do j=1,jj
do i=1,ians2
read(13,*)
i1=i1+1
end do
if(ians1.ne.0)then
```

```

read(13,*)
i1=i1+1
end if
end do
end do
end do
end do
i1=0
write(*,*)i1
do itstep=6,100000000
write(*,*)itstep
read(13,*,end=26)
read(13,*)
read(13,*)
i1=i1+3
do nzone=1,46
read(13,11,end=26)zone,ii,jj
read(13,*)
read(13,*)
i1=i1+3
if(itstep.eq.(6+5*i1))then
write(14,20)zone,ii,jj
20 format('ZONE T="K=',I7,', ', I= ',I7,', J= ',I7,',
F=BLOCK ')
write(14,*)'DT=(SINGLE SINGLE SINGLE SINGLE SINGLE SINGLE SING
LE SINGLE SINGLE SINGLE SINGLE)'
end if
if ((mod(ii,5)).eq.0)then
ians1=0
ians2=ii/5
else
ians1=mod(ii,5)
ians2=(ii-ians1)/5
end if
do nvar=1,11
do j=1,jj
do i=1,ians2
! write(*,*)itstep
! write(*,*)"reached here"
! write(*,*)i1
read(13,21)v1,v2,v3,v4,v5
if(itstep.eq.(6+5*i1))then

```

```

write(14,21)v1,v2,v3,v4,v5
end if
end do
if(ians1.ne.0)then
if(ians1.eq.1)then
read(13,22)v6
if(itstep.eq.(6+5*in1))then
write(14,22)v6
end if
end if
if(ians1.eq.2)then
read(13,23)v6,v7
if(itstep.eq.(6+5*in1))then
write(14,23)v6,v7
end if
end if
if(ians1.eq.3)then
read(13,24)v6,v7,v8
if(itstep.eq.(6+5*in1))then
write(14,24)v6,v7,v8
end if
end if
if(ians.eq.4)then
read(13,25)v6,v7,v8,v9
if(itstep.eq.(6+5*in1))then
write(14,25)v6,v7,v8,v9
end if
end if
end if
end do
end do
end do
iin1=iin1+1
end do
11 format(13X,I2,9X,I2,7X,I2)
21 format(5(1x,E15.8))
22 format(1x,E15.8)
23 format(2(1x,E15.8))
24 format(3(1x,E15.8))
25 format(4(1x,E15.8))
26 continue

```

end program