

CRANFIELD UNIVERSITY

Faculty of Engineering and Applied Sciences

M.Sc. Thesis

Academic Year: 2024-2025

Thomas Stiley

Investigation into the modelling ability of a hybrid
anisotropic K-omega SST/STM for the Onera M6 wing and
DNS lab channel flow validation cases

Supervisor(s):

Dr. László Könözy

© Cranfield University, 2025.

All rights reserved. No part of this publication may be reproduced
without the written permission of the copyright holder.

Except where acknowledged in the customary manner, the material presented in this thesis is, to the best of my knowledge, original and has not been submitted in whole or part for a degree in any university.

Thomas Stiley

Abstract

The chosen turbulence model for a RANS based computational fluid dynamics simulation is crucial for accurate CFD results. Many strong models exist such as the K-omega SST, K-epsilon and Spalart Allmaras turbulence models however, each of these models makes a major assumption which in nature is profoundly incorrect. Most commercially available turbulence models assume that the Reynolds stress tensor is isotropic, when in reality any shear, rotational or wall bounded flows are anisotropic in nature. This paper aims to investigate a newly proposed anisotropic hybrid K-omega SST / stochastic turbulence model developed by Könözy (1)(2). The model has been tested for two validation cases: an Onera M6 transonic test case and a traditional channel flow case. Data has been plotted and compared to experimental data and DNS data and good alignment was found for both cases between the two data sets. The model outperforms the K-omega SST in its physical modelling however does still need some refinement in its model constants approach as shown by its anisotropic Reynolds stress modelling for the channel flow case.

Acknowledgements

I would like to acknowledge the guidance from my supervisor Dr László Könözy with regards to implementing his K-omega SST/STM hybrid anisotropic turbulence model as well as his overall help with this project.

I would like to state that chat GPT 5 was used in the formatting of latex tables and figures.

Contents

Abstract	v
Acknowledgements	vi
List of Figures	xi
List of Tables	xii
1 Introduction	1
1.0.1 Turbulence modelling	1
1.0.2 Aims and Objectives	2
2 Literature review	3
2.1 Navier Stokes equations	3
2.1.1 Large Eddy simulations	5
2.1.2 Detached eddy simulations	6
2.2 Reynolds Averaged Navier Stokes equations	6
2.3 Closure models for the Reynolds stress tensor	8
2.3.1 Algebraic models	8
2.3.2 One or two equation closure models	10
2.3.2.1 K-epsilon family of turbulence models	10
2.3.2.2 K-Omega family of turbulence models	11
2.3.2.3 Spalart Allmaras model	13
2.3.3 Reynolds stress models	14
2.4 Anisotropy of turbulent flows	15
2.4.1 RANS anisotropic turbulence models	15
2.4.2 The hybrid K-omega SST Stochastic turbulence model	16
3 Methodology	19
3.1 Onera M6 wing case	19
3.1.1 Domain	20

Contents

3.1.2	Simulation set up	20
3.1.3	Mesh	22
3.1.4	Grid study	23
3.2	Channel flow case	25
3.3	Domain	26
3.4	Simulation set up	27
3.4.1	Mesh	30
3.5	Mesh study	31
3.5.1	DNS lab case	32
4	Results and Discussion	36
4.1	Onera M6 validation case	36
4.1.1	Shear stress and Shock capturing	36
4.1.2	Turbulence capturing	42
4.1.3	Pressure coefficients and absolute error	44
4.1.3.1	Mesh analysis	44
4.1.3.2	Turbulence model comparisons	49
4.2	Channel flow validation case	54
4.2.1	Reynolds stresses comparisons	55
4.2.2	Supporting contours comparisons	64
4.2.2.1	Turbulence capturing	64
4.2.2.2	Velocity contours	69
4.3	Conclusion and Future Work	71
4.3.1	Conclusion	71
4.3.2	Future Work	71
	References	73
4.4	Appendices	80
	Appendices	80
4.4.1	Appendix A:	80

List of Figures

3.1	Domain for the Onera M6 case	20
3.2	Original mesh of the Onera M6 case	23
3.3	Adaptive mesh refinement for density gradient	23
3.4	Drag coefficient GCI study for the Onera M6 wing case	24
3.5	Lift coefficient GCI study for the Onera M6 wing case	25
3.6	Domain used for the channel flow simulations	27
3.7	X-Shear stress plot along the lower wall of the domain	28
3.8	Turbulence intensity study	28
3.9	Channel flow Grid convergence study	31
3.10	Enhanced view of the inflation layers of the finest channel flow mesh . . .	32
4.1		
	(a) Mach number contours for mesh 1 Anisotropic model	
	(b) Mach number contours for mesh 2 Anisotropic model	
	(c) Mach number contours for mesh 3 Anisotropic model	
	(d) Mach number contours for mesh 2 K-omega SST model	38
4.2		
	(a) Shear stress contours for mesh 1 Anisotropic model	
	(b) Shear stress contours for mesh 2 Anisotropic model	
	(c) Shear stress contours for mesh 3 Anisotropic model	
	(d) Shear stress contours for mesh 2 K-omega SST model	40
4.3		
	(a) Velocity contours on Q-criterion iso-surface for mesh 1 Anisotropic model	
	(b) Velocity contours on Q-criterion iso-surface for mesh 2 Anisotropic model	
	(c) Velocity contours on Q-criterion iso-surface for mesh 3 Anisotropic model	
	(d) Velocity contours on Q-criterion iso-surface for mesh 2 K-omega SST model	43

List of Figures

4.4	Comparison of mesh performance at capturing the pressure coefficient compared to experimental data	46
4.5	(a) Norm 1 comparison of errors from the three meshes (b) Norm 2 comparison of errors from the three meshes (c) Norm max comparison of errors from the three meshes	47
4.6	Comparison of turbulence model performance at capturing the pressure coefficient compared to experimental data	50
4.7	(a) Norm 1 comparison of errors from the turbulence models (b) Norm 2 comparison of errors from the turbulence models (c) Norm max comparison of errors from the turbulence models	53
4.8	(a) Comparison of non dimensional RMS 11 Reynolds stress for channel flow (b) Comparison of non dimensional RMS 22 Reynolds stress for channel flow (c) Comparison of non dimensional RMS 33 Reynolds stress for channel flow 57	
4.9	(a) Comparison of non dimensional RMS 12 Reynolds stress for channel flow (b) Comparison of non dimensional RMS 13 Reynolds stress for channel flow (c) Comparison of non dimensional RMS 23 Reynolds stress for channel flow 59	
4.10	(a) Comparison of non dimensional RMS 11 Reynolds stress for channel flow (b) Comparison of non dimensional RMS 22 Reynolds stress for channel flow (c) Comparison of non dimensional RMS 33 Reynolds stress for channel flow 61	

List of Figures

4.11

- (a) Comparison of non dimensional RMS 12 Reynolds stress for channel flow
 - (b) Comparison of non dimensional RMS 13 Reynolds stress for channel flow
 - (c) Comparison of non dimensional RMS 23 Reynolds stress for channel flow
- 63

4.12

- (a) y plus contour for mesh 1
 - (b) y plus contour for mesh 2
 - (c) y plus contour for mesh 3
- 65

4.13

- (a) Turbulent viscosity contour on Q-criterion iso-surface for mesh 1
- (b) Turbulent viscosity contour on Q-criterion iso-surface for mesh 2
- (c) Turbulent viscosity contour on Q-criterion iso-surface for mesh 3
- (d) Turbulent viscosity contour on Q-criterion iso-surface for mesh 3 with adjusted constants 68

4.14

- (a) y plus contour for mesh 1
 - (b) y plus contour for mesh 2
 - (c) y plus contour for mesh 3
 - (d) y plus contour for mesh 3 with adjusted constants
- 70

List of Tables

3.1	Flow and geometry parameters using engineering notation.	22
3.2	ONERA M6 wing mesh study	24
3.3	Channel flow mesh study	30
3.4	Mesh quality of the finest channel flow mesh.	32

List of Equations

2.1	Navier Stokes conservation of mass equation	4
2.2	Navier Stokes conservation of momentum equation	4
2.3	Navier Stokes conservation of energy equation	4
2.4	Reynolds decomposition	7
2.5	Reynolds momentum equation	7
2.6	Reynolds stress tensor (1)	7
2.7	Boussinesq eddy viscosity model (3)	9
2.8	Prandtl mixing length model (3)	9
2.9	Prandtl wake mixing length model (3)	9
2.10	Boussinesq eddy viscosity model (3)	9
2.11	Generalised Boussinesq hypothesis for incompressible flows	10
2.12	Standard k-epsilon Turbulent kinetic energy transport equation	10
2.13	Standard k-epsilon Turbulent dissipation rate transport equation	11
2.14	Standard k-epsilon Production term	11
2.15	Standard k-epsilon turbulent viscosity equation	11
2.16	Standard k-omega Turbulent kinetic energy transport equation	12
2.17	Standard k-omega Specific Dissipation rate transport equation	12
2.18	k-omega SST Turbulent kinetic energy transport equation	12
2.19	k-omega family turbulent eddy viscosity formulations	12
2.20	k-omega family Turbulent kinetic energy production term formulations	13
2.21	k-omega family Diffusive constant formulations	13
2.22	k-omega SST Specific Dissipation rate transport equation	13
2.23	Spalart Allmaras modified turbulent viscosity transport equation	14
2.24	Reynolds stress transport equation	15
2.25	Reynolds stress epsilon transport equation	15
2.26	Hybrid k-omega SST/STM Reynolds stress tensor formulation (1)	17
2.27	Hybrid k-omega SST/STM Specific dissipation rate transport equation (1)	17
2.28	Hybrid k-omega SST/STM Turbulent kinetic energy transport equation (1)	17
2.29	Principle turbulent shear stress equation (1)	17
3.1	Mach number equation	21
3.2	Reynolds number equation	21

List of Equations

3.3	Temperature relation equation	21
3.4	New μ equation	21
3.5	Friction velocity equation	29
3.6	Friction Reynolds number equation	29
3.7	Non dimensional RMS Reynolds stress tensor component 11	29
3.8	Non dimensional RMS Reynolds stress tensor component 22	29
3.9	Non dimensional RMS Reynolds stress tensor component 33	29
3.10	Non dimensional RMS Reynolds stress tensor component 12	30
3.11	Non dimensional RMS Reynolds stress tensor component 13	30
3.12	Non dimensional RMS Reynolds stress tensor component 23	30
3.13	Non dimensional RMS Reynolds stress tensor component from DNS lab 11 .	33
3.14	Non dimensional RMS Reynolds stress tensor component from DNS lab 22 .	33
3.15	Non dimensional RMS Reynolds stress tensor component from DNS lab 33 .	34
3.16	Anisotropic fluctuation calculation from DNSlab	34
3.17	Non dimensional RMS Reynolds stress tensor component from DNS lab 12 .	34
3.18	Non dimensional RMS Reynolds stress tensor component from DNS lab 13 .	34
3.19	Non dimensional RMS Reynolds stress tensor component from DNS lab 23 .	34

Nomenclature

$\bar{\bar{\tau}}$	Viscous stress tensor
α	Incidence
α	Model constant
$\bar{\bar{\tau}}_R$	Reynolds stress tensor
$\bar{\bar{G}}$	Transformed dimensionless deviatoric similarity tensor
$\bar{\bar{I}}$	Identity tensor
$\bar{\bar{S}}$	Mean strain rate tensor
$\bar{u}$	Mean velocity
$\bar{u}$	Velocity
β	Model constant
β^*	Model constant
δ	Half channel height
ε	Turbulent Dissipation rate
ε	Turbulent dissipation rate
ε_{ij}	Dissipation tensor
γ	Function of Menter's blending function
γ	Ratio of specific heats
μ	Viscosity
μ_Θ	Dimensionless scalar weight parameter

Nomenclature

μ_t	Turbulent viscosity
ν	Kinematic viscosity
ν_t	Eddy viscosity
ν_t	Turbulent kinematic viscosity
ω	Specific dissipation rate
Φ_{ij}	Pressure-strain correlation
ρ	Density
σ	Diffusion constant
σ_ε	Model constant for diffusion
σ_ε	Model constant
σ_ω	Diffusion coefficient
σ_k	Diffusion coefficient
σ_k	Model constant
σ_R	Model constant
σ_{k1}	Model constant
σ_{k2}	Model constant
Θ	Principal dominant turbulent shear stress
$\tilde{\nu}$	Modified kinematic viscosity
a_1	Model constant
b	Wing span
C_μ	Model constant
C_D	Drag coefficient
C_L	Lift coefficient
C_P	Pressure coefficient

Nomenclature

$C_{1\varepsilon}$	Model constant
$C_{2\varepsilon}$	Model constant
$C_{\varepsilon 1}$	Model constant
$C_{\varepsilon 2}$	Model constant
C_{b1}	Production constant
C_{b2}	Non-linear diffusion constant
c_{MAC}	Mean Aerodynamic Chord
C_{w1}	Wall-destruction constant
d	Wall distance
D_h	Hydraulic diameter
e_t	total specific energy
f	body-force vector
F_1	Primary SST blending function
F_2	SST Blending function
f_w	Wall damping function
f_{t2}	Damping function
g	Gravitational acceleration
I	Turbulence intensity
k	Empirical constant
k	Von Karmen constant
l	Mixing length
M	Mach number
P	Production term
p	Pressure

Nomenclature

p'	Fluctuating pressure component
P_K	Production term of turbulent kinetic energy
p_T	Total pressure
P_{ij}	Production term
q	heat flux
R	Gas constant
R_{ij}	Reynolds stress tensor component
Re	Reynolds number
Re_τ	Friction Reynolds number
RMS	Root mean square value
S	Wing area
$S_{ij}S_{ij}$	Second invariant of the strain rate tensor
T	Temperature
t	Time
U	Mean stream wise velocity
u'	Fluctuating velocity component
u'	Stream wise fluctuations
u'_1	Stream wise velocity fluctuations
u'_2	wall-normal velocity fluctuations
u'_3	Span wise velocity fluctuations
U_∞	Free stream velocity
u_τ	Friction velocity
u_t	Total velocity
u_x	stream wise velocity component

Nomenclature

u_y	Vertical velocity component
v	Wall-normal velocity fluctuations
v'	Wall normal fluctuations
w'	Spanwise fluctuations
x/c	Non-dimensional chordwise position
X	Characteristic length scale of turbulence
Y	Wall-normal coordinate
y^+	Dimensionless wall distance

Chapter 1

Introduction

1.0.1 Turbulence modelling

The Reynolds averaged Navier stokes equations continue to be a cornerstone of modern fluid dynamics work. The method shows significant computational cost savings over alternative methods like LES and DNS while maintaining strong correlation between experimental and simulated data. This computational cost saving is crucial for industrial applications of fluid dynamic solvers as it allows for rapid development and optimisation, aided by the ever increasing computational power of modern day hardware. However, the RANS and URANS equations have one major issue for their implementation. This comes from the lack of closure without use of a turbulence model to calculate the Reynolds stress tensor. Many models have been tested and validated against experimental data to be reasonably accurate, in particular the k-omega SST model offers a high degree of accuracy while maintaining a low computational cost. Other one or two equation eddy-viscosity turbulence models such as Spalart-Allmaras and k-epsilon can also produce reasonable results, but are more sensitive to the application and assume the Reynolds stress tensor so turbulence to be isotropic. Other solutions like the Reynolds stress models provide increased accuracy over the eddy-viscosity turbulence models however, come with a much higher computational cost with seven additional transport equations for three dimensional flows. However, these models take into account the anisotropy of turbulence in the Reynolds stress tensor by modelling each Reynolds stress individually therefore, they can be considered to be the most physically correct of the RANS models discussed. This paper aims to add validation of a recently developed anisotropic hybrid k-omega SST / Stochastic turbulence model developed by Dr Könözy (1)(2) as well as investigate the implementation of an adapted set of equations to better model the energy balance of a turbulent boundary layer.

1. Introduction

1.0.2 Aims and Objectives

- Analyse the turbulence models performance through shocks in the Onera M6 wing transonic flow case
- Discuss the effects of the adaptive mesh refinement strategy and any issues with the turbulence model using this methodology
- Compare the anisotropic model to the K-omega SST for the transonic flow case.
- Analyse the turbulence model's performance in a low Reynolds number case for the channel flow
- Investigate any improvements in near-wall modelling which can be made by changing the model's constants
- Assess the nature of the model with regards to grid sensitivity and if good results can be obtained with a coarser mesh

Chapter 2

Literature review

Many see turbulence as the last unsolved phenomena of classical physics and so much research has been performed in the area since as early as the 19th century (4). Turbulence modelling therefore, is crucial to any engineering application involving complex flow whether it be rotational, high curvature or supersonic flow regimes. Much research has been undertaken into turbulent structures in flow and its behaviours however, the full story of turbulent flows is yet to be discovered. Literature provides many methods in which flow cases can be modelled to gain insight to the fluids behaviour, each having their own benefits and drawbacks. Direct numerical simulations of the Navier-stokes equations form the fundamental model of any flow case, resolving the smallest to the largest scales of turbulence. LES simulations provide a solution to turbulence on the larger length scales and use sub grid models for the capturing of small scale turbulence. RANS and URANS model fluid cases using averaged instantaneous values with fluctuations modelled through the Reynolds stress tensor. However, this set of equations require additional closure before implementation. This is where engineering and turbulence models are crucial. This literature will look to give a broad view of what is being researched as well as the fundamentals of computational fluid dynamics methods. In addition, the subject of this paper will be explored to analyse areas which need to be explored where gaps in the literature can be identified.

2.1 Navier Stokes equations

In the early 19th century, Claude-Louis Navier attempted to derive the governing laws of fluid motion using an extension of Euler's equations by composing fluids of molecules subject to internal forces. Navier's work was the first to incorporate viscosity into the fluid motion equations (5). Later, George Gabriel Stokes refined Navier's equations of fluidic

2. Literature review

motion using continuum mechanics and assumptions with an isotropic viscosity tensor and for the fluids to be Newtonian. This work was laid out in Stoke's paper in 1845 (6), these two pieces of work form the foundations of the modern Navier Stokes equations, fundamental to any fluid dynamics problems seen in equations 2.1, 2.3 and 2.2.

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{u}) = 0 \quad (2.1)$$

$$\frac{\partial \rho \bar{u}}{\partial t} + \nabla \cdot (\rho \bar{u} \otimes \bar{u}) = \rho g - \nabla p + \nabla \cdot \bar{\tau} \quad (2.2)$$

$$\frac{\partial (\rho e_t)}{\partial t} + \nabla \cdot (\rho e_t \mathbf{u}) = -\nabla \cdot (p \mathbf{u}) + \nabla \cdot (\boldsymbol{\tau} \cdot \mathbf{u}) - \nabla \cdot \mathbf{q} + \rho \mathbf{f} \cdot \mathbf{u} \quad (2.3)$$

The Navier Stokes equations can be used to form a fundamental view of fluid dynamics however, their computation for flow cases remains a severe challenge. Direct numerical simulation's can be used to solve the Navier Stokes equations in their full form however, computational cost severely limits the size and volume of cases solved using this method. This computational cost can mean some solutions are unobtainable with current technology levels for a full aircraft (4). This is largely due to the number of grid cells growing with Reynolds number due to the greater range of Kolmogorov scales present. The DNS simulations must capture each scale and so only Reynolds numbers up to 10^5 seem to be feasible at current technology levels (7). Some consideration should be made to environmental, as well as, computational cost of DNS computation. A study by Pascal Domingo and Luc Vervisch estimates the carbon cost of running a DNS case with approximately 2 Billion cell to be approximately 776KG of CO_2 emissions (8).

Improved meshing strategies (7) and an increase in computational power may increase the feasibility of more advanced DNS studies. In addition, quantum computing may offer some solutions. Some research shows that 2D cases using the Lattice Boltzmann method show good qualitative correlation between the quantum and original algorithms for solving a classical flow around a cylinder test case (9). However, a paper by Syamlal and others shows current quantum algorithms seem to be slower than their conventional

2. Literature review

counterparts (10). The same paper does seem encouraging and suggest some future improvements may get this computational time reduced significantly, but it can be concluded that this technology seems to be one of the future.

2.1.1 Large Eddy simulations

A strong alternative for direct numerical simulations comes from the LES model. The LES model was first successfully applied to a three dimensional flow case by Deardoff in 1970 (11). This study built on the foundational work of Lilly (12) and Smargorinsky (13). The model was originally formulated for applications in atmospherical sciences (14). LES models work by filtering the time dependant Navier Stokes equations to determine turbulent eddies below the grid scale (15). Sub grid models can be used to determine the nature of the eddies below the grid size. Eddies larger than or in the same scale as the grid can be solved explicitly. LES models offer significantly reduced computational cost, but also reduced accuracy over DNS studies. Some studies propose new solutions to the sub grid stress tensors (16) to improve accuracy. Probably the biggest downfall for LES models comes from its high computational costs making it unpractical for engineering flows. This is exacerbated by the LES model performing best in high Reynolds number simulations where computational cost is at its peak (17).

Some advancements in reducing computational cost while maintaining a high degree of accuracy come through the use of hybrid models. Balaras and others suggested a two layer approach to boundary layer modelling, with a wall model used for the log-law region and a standard LES model for the outer wall regions (18). This wall modelled hybrid LES model was further refined by Cabot in 2000 (19) to produce modern WMLES models. The computational cost savings from WMLES come from the near wall region where an extremely fine grid would be needed for traditional LES. Therefore, using a wall modelling approach for this region, which requires a much coarser mesh, reduces the cell count required dramatically. Cost savings were estimated by Yang, Griffin and Fu in papers between 2020 and 2021 (20) (21). These papers assume computational cost can be represented by an exponent of the Reynolds number. Their findings assessed multiple solving methods for their costs DNS was estimated to be $Re^{2.91}$, LES $Re^{2.72}$ and WMLES to be $Re^{1.14}$ (20). These values show significant improvements in computational cost using the new wall model proposed in (21).

2. Literature review

2.1.2 Detached eddy simulations

Detached eddy simulations are an alternative hybrid approach to flow prediction. Similarly to WMLES, DES does not use LES for the near wall region, instead using a modified RANS turbulence model to predict separation before transitioning to an LES approach for separated regions. The model was first suggested in 1977 by Spalart et al, using a modified version of the Spalart Allmaras turbulence mode (22) (23). The modification changes the destructive term for the eddy viscosity to scale the wall distance to be used to switch between Spalart-Allmaras and Sub grid scale turbulence modelling (22). This allows for LES to be used when sub grid scale models are active to resolve the large eddies present with vortex shedding in separated flow regions. DES was designed for use in engineering flows for aerospace applications however, it is highly sensitive to boundary layer grid generation in high thickness boundary layers or shallow separation bubbles (24).

Other alternatives exist for hybridised LES-RANS however, an accurate and highly robust model is still yet to be fully validated. Continuous eddy simulations offers some hope in this area. This method is characterised by its minimal error simulation methods, making it stand out from WMLES and DES simulations (25). In addition, modifications the DES model have been proposed. DDES or Delayed Detached Eddy simulations offer an alteration to the original DES97 formulation (22) to account for grid ambiguity as mentioned previously. This works to delay the switch to LES in non-detached flow regions (26). In addition IDDES or Improved Delayed Detached eddy simulations works using a blending function of DDES and WMLES. This is dependant on the level of flow separation and if inflow turbulence content is present. The blending is outlined in the paper by Shur et al. from 2008 (27). These models both improve upon the original formulations but add large levels of complexity for implantation and further adaptations. In addition, the increase in computational cost over pure RANS methods is still present.

2.2 Reynolds Averaged Navier Stokes equations

The RANS equations have been fundamental for engineering flows for decades and act as a lower computational cost alternative to LES and DNS methods. However, this reduction in computational cost comes with a significant decrease in accuracy due to it's averaging of complex unsteady flows and reliance on often non-physically formulated turbulence models.

The Reynolds averaged Navier Stokes equations were originally formulated by Osborne Reynolds in 1895 (28). Reynolds proposed a decomposition of the instantaneous

2. Literature review

velocity field into its mean and fluctuating terms. The Reynolds decomposition allows for a separation between the steady mean flow variables and the fluctuations of these variables due to turbulent structure in the flow. This decomposition can be seen in equation 2.4. The decomposed flow variables can be implemented into the Navier Stokes momentum equation seen in equation 2.2. Reynolds proposed an averaging technique to eliminate some variables in the Navier Stokes equations with the logical rules that any averaged fluctuations will be zero and that the averaging of two product flow variables can be decomposed into the product of their mean values and the mean product of their fluctuations. Following this, the Reynolds averaged Navier Stokes equations can be formulated with the momentum equation seen in equation 2.5.

The Reynolds averaged momentum conservation equation serves as a different approach to modelling turbulence compared to its direct solution through the Navier Stokes equations. However, this approach has a closure issue. The three momentum equations, continuity and energy equations act to close all unknowns in the PDE apart from the Reynolds stress tensor, denoted by $\bar{\tau}_R$. The closure of the RANS equations is the reason turbulence models are so crucial for the analysis of complex engineering flows.

$$\bar{u}_T = \bar{u} + u' \quad p_T = p + p' \quad (2.4)$$

$$\frac{\partial(\rho \bar{u})}{\partial t} + \nabla(\rho \bar{u} \otimes \bar{u}) = \rho g - \nabla p + \mu \nabla^2 \bar{u} + \nabla \bar{\tau}_R \quad (2.5)$$

$$\bar{\tau}_R = -\rho \overline{u' u'} = \begin{bmatrix} \overline{u'_1 u'_1} & \overline{u'_1 u'_2} & \overline{u'_1 u'_3} \\ \overline{u'_2 u'_1} & \overline{u'_2 u'_2} & \overline{u'_2 u'_3} \\ \overline{u'_3 u'_1} & \overline{u'_3 u'_2} & \overline{u'_3 u'_3} \end{bmatrix} \quad (2.6)$$

The Reynolds stress tensor holds the key components of turbulent flows. It is comprised of the Reynolds stresses for each component of the spatial field. However, unlike the other unknowns presented from the momentum equation, there are no new equations which can be gained to solve for the Reynolds stress tensor directly (29). By inspection, the Reynolds stress tensor is symmetrical in nature about its three diagonal elements (29).

2. Literature review

From equation 2.6, the diagonal elements can be identified and the symmetry shows that $\overline{u'_1 u'_2} = \overline{u'_2 u'_1}$, $\overline{u'_1 u'_3} = \overline{u'_3 u'_1}$ and $\overline{u'_2 u'_3} = \overline{u'_3 u'_2}$. This presents six unknown Reynolds stresses which must be solved.

2.3 Closure models for the Reynolds stress tensor

The closure of the stress tensor is vital for modelling any RANS flow case and many methods for closing the tensor have been proposed over the years. The most commonly used method for engineering flows relies on the Boussinesq equation for representing the Reynolds stresses in the tensor. This equation forms the foundation of the one or two equation eddy viscosity equations which are the cornerstone of industrial computational fluid dynamics (1). Another category can be derived as first order closure models, relying on functions of the mean flow. A different approach to closure can be derived from the Reynolds stress transport models which provide a more physical solution. Finally some recent focus has been targeted at anisotropic turbulence modelling. These models allow for a more physical representation of turbulent flows.

A turbulence model can be assessed for suitability based on a number of parameters. Computational cost must be low for application to industrial flow problems using RANS. A steep increase in cost would allow for less benefit over more advanced LES or RANS hybrid models. Accuracy is evidently a crucial indicator of a turbulence models ability. In addition, adaptability to a variety of flow cases is key for a turbulence models success. This can be extended to sensitivity of the model to flow conditions and mesh settings. With regards to accuracy, turbulence models tend to struggle with predicting separation and modelling Reynolds stresses correctly in separated regions (30). It is perhaps the fatal flaw in RANS that the oversimplification of the non-linear Navier-stokes equations leads to the approximate and largely un-physical nature of its closure models(3).

2.3.1 Algebraic models

Literature has presented a number of methods of closure. Zero-equation algebraic closures were among the first turbulence models proposed and implemented. Zero-equation models represent the tensor using a function of the mean-flow (3). Examples of these models can be seen in equations 2.7, 2.8, 2.9 and 2.10. The algebraic models listed below are early models however, some improvements have been proposed. Obermeier proposed an adaption to the characteristic length used in Prandtl's mixing layer model. Instead of

2. Literature review

using a proportional relationship between the the fluctuations in u and v , a function is applied to tune the fluctuations in the near wall region (31). This modification improved results for the model considerably for a range of Reynolds numbers.

Other models, such as the zero-equation Algebraic model proposed by Van Driest (32) show better performance in some cases when compared to a k-epsilon model. This was shown in a paper by Bazargan and Mohseni where the Van Driest model and k-epsilon model were tested for different bouyency forces in supersonic flows (33).

$$-\overline{uv} = \nu_t \frac{\partial U}{\partial Y} \quad (2.7)$$

$$-\overline{uv} = l^2 \frac{dU}{dY} \frac{\partial U}{\partial Y} \quad (2.8)$$

$$-\overline{uv} = kX \frac{\partial U}{\partial Y} \quad (2.9)$$

$$-\overline{uv} = k^2 \left[\frac{dU}{dY} / \frac{d^2U}{dY^2} \right]^2 \frac{\partial U}{\partial Y} \quad (2.10)$$

Ultimately, though the algebraic models are gross oversimplifications of the flow problems which lend themselves to particular applications (3), they struggle outside of specific cases. These models were mainly derived for use in simple flow problems in a time before computational power could solve first and second order closure methods and so their applicability is limited.

2. Literature review

2.3.2 One or two equation closure models

One or two equation models such as Spalart-Allmaras, k-epsilon and the k-omega models are the backbone of turbulence modelling for engineering applications. All models vary in their number of equations and formulation however, they share mostly the same structure, with each being a new transport equation for turbulent flow parameters. The models are all based on the Generalised Bousinesq hypothesis seen in its incompressible form in equation 2.11 (34). This equation represents the Reynolds stress tensor as an isotropic combination of the turbulent pressure effects and internal stresses. The turbulent kinetic energy is solved in most one or two order equations through its own transport equation. The Turbulent viscosity term is solved uniquely to each turbulence model, usually as a function of various flow properties calculated through additional transport equations. The first to implement this approach to RANS was Prandtl in 1925 (35), though this was using zero-equation algebraic models previously discussed.

$$\bar{\tau}_R = 2\mu_t \bar{S} - \frac{2}{3}\rho k \bar{I} \quad (2.11)$$

2.3.2.1 K-epsilon family of turbulence models

The two equation k-epsilon turbulence model was outlined by Launder and Spalding in 1974 (36). The model uses two separate transport equations as seen in equations 2.13 and 2.12. The method uses the transport of turbulent kinetic energy and turbulent dissipation rate to model the turbulent eddy viscosity for the Bousinesq equation. The k-epsilon method like other turbulence models uses production, dissipation and diffusion terms to represent the energy balance of the turbulent flow. The production term can be seen in equation 2.14 and the turbulent eddy kinematic viscosity in equation 2.15.

$$\frac{\partial k}{\partial t} + \vec{u} \cdot \nabla k = \nabla \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \nabla k \right] + P_k - \epsilon \quad (2.12)$$

$$\frac{\partial \epsilon}{\partial t} + \vec{u} \cdot \nabla \epsilon = \nabla \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_\epsilon} \right) \nabla \epsilon \right] + C_{1\epsilon} \frac{\epsilon}{k} P_k - C_{2\epsilon} \frac{\epsilon^2}{k} \quad (2.13)$$

2. Literature review

The k-epsilon model has a wide variety of applications largely for industrial flows for pipes, channels, and some aerospace applications. The k-epsilon model is one of the oldest two equation models and so its application and validity has been validated over a wide variety of cases. The model seems to have particularly good alignment with experimental data in free shear flows. This was studied by Launder in 1973 (37), where several models were compared including a k-epsilon model, mixing length models and an RSM model. The k-epsilon method was found to outperform the mixing length models but fall short of the RSM results (37). This can be expected and the high computational cost of Reynolds stress models must be taken into account when analysing these results.

$$P_K = 2\nu_t S_{ij} S_{ij} \quad (2.14)$$

A fundamental flaw with the k-epsilon model is its inherent isotropic nature. This makes it unsuited to highly detached and swirling flows. A study by Turutoglu and Cadirci found that some improvements can be made by adjusting closure coefficients for highly anisotropic flow cases like a round free jet flow (38). In addition, Chen and Kim presented findings comparing the standard k-epsilon model to an adapted production term k-epsilon model (39). This study indicates that again in the highly anisotropic confined swirling flow and backward facing step cases, the standard model performs poorly but the presented adaptations may improve results significantly (39). The k-epsilon model overall shows good robustness for a variety of cases but its applications to anisotropic aerospace flows may be limited.

$$\nu_t = C_\mu \frac{k^2}{\epsilon} \quad (2.15)$$

2.3.2.2 K-Omega family of turbulence models

The K-omega family, more specifically the K-omega SST model remains one of the most widely used turbulence models for aerospace and general applications. The K-omega model was originally proposed by Willcox in 1988 (40). The model uses the turbulent kinetic energy similarly to the k-epsilon model however, the two models differ in their

2. Literature review

second equation. While the k-epsilon model uses turbulent dissipation rate, the K-omega model uses the specific dissipation rate. The two transport equations for the base model can be seen in equations 2.16 2.17.

$$\frac{\partial k}{\partial t} + \vec{u} \cdot \nabla k = \nabla \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \nabla k \right] + P_k - \beta^* k \omega \quad (2.16)$$

$$\frac{\partial \omega}{\partial t} + \vec{u} \cdot \nabla \omega = \nabla \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_\omega} \right) \nabla \omega \right] + \alpha \frac{\omega}{k} P_k - \beta \omega^2 \quad (2.17)$$

The K-omega family fundamentally improves on near wall modelling when compared to the K-epsilon model however, it is sensitive in free stream flows. Whereas, the Standard k-epsilon model shows to be sensitive in the viscous sublevel/buffer layers but performs well for free stream flow (41). This led to the development of the K-omega SST turbulence model by Menter in 1994 (42). This model aimed to blend the K-omega and K-epsilon models depending on the wall distance, its transport equations can be seen in equations: 2.18 and 2.22. It can be noted that the K-omega standard model and the SST model both maintain a nearly identical looking turbulent kinetic energy transport equation. However, the formulation of the turbulent eddy viscosity differs between the two models. In addition, the formulation of the production term P_k and diffusive constant σ_k change between the models. These alterations can be seen in equations: 2.19, 2.20 and 2.21.

$$\frac{\partial k}{\partial t} + \vec{u} \cdot \nabla k = \nabla \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \nabla k \right] + P_k - \beta^* k \omega \quad (2.18)$$

$$\begin{aligned} \nu_t &= \frac{k}{\omega} & \text{Standard } k-\omega \\ \nu_t &= \frac{a_1 k}{\max(a_1 \omega, SF_2)} & k-\omega \text{ SST} \end{aligned} \quad (2.19)$$

2. Literature review

$$\begin{aligned} P_k &= \nu_t S^2 & \text{Standard } k - \omega \\ P_k &= \min(\nu_t S, 10\beta^* k \omega) & k - \omega \text{ SST} \end{aligned} \quad (2.20)$$

The K-omega SST model improves on the original model in most applications due to the free stream flow improvements from the k-epsilon model. In addition, it seems to handle adverse pressure gradients well, this being a common issue for most models (43). Studies also show the model to perform well in attached flow and also in fully stalled flow however, note that the model struggles in modelling angles where stall begins to develop (44). This is further reinforced in a paper studying the models performance modelling wind turbine blades with significant wall roughness. It was found that the model struggled with modelling separation location and over-predicts the Reynolds stresses near the surface (45). Overall, the issues listed above can generally be attributed to struggles with modelling complex flows using the RANS equations and may not be entirely turbulence model related. In general K-omega SST performs extremely well for most turbulent flows (46), hence justifying its use for the hybrid anisotropic K-omega SST/STM model.

$$\begin{aligned} \sigma_k &= 2.0 & \text{Standard } k - \omega \\ \sigma_k &= F_1 \cdot \sigma_{k1} + (1 - F_1) \cdot \sigma_{k2} & k - \omega \text{ SST} \end{aligned} \quad (2.21)$$

$$\frac{\partial \omega}{\partial t} + \vec{u} \cdot \nabla \omega = \nabla \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_\omega} \right) \nabla \omega \right] + \alpha \frac{\omega}{k} P_k - \beta \omega^2 + 2(1 - F_1) \frac{\sigma_{\omega 2}}{\omega} \nabla k \cdot \nabla \omega \quad (2.22)$$

2.3.2.3 Spalart Allmaras model

The Spalart Allmaras eddy viscosity turbulence model is rather unique in its field with the model consists of one transport equation in its formulation. This equation models the transport of modified turbulent viscosity and was originally proposed By Spalart and Allmaras in 1992 (23). With this turbulence model consisting of only one equation, it has low computational cost while still being considered highly accurate. This has led to it being among the most popular turbulence model for aerospace applications along with K-omega SST. The model and its coefficients were originally formulated and optimised for aerospace applications by Spalart and Allmaras (23). The model is therefore best suited for aerodynamic / turbo machinery applications where there is mild flow sep-

2. Literature review

aration present as well as compressible effects from supersonic and transonic flows (47). This model is anisotropic in nature like most commercially available turbulence models and so inherently the model may struggle with representing physical near-wall turbulence.

The transport equation for the Spalart-Allmaras model can be seen in equation 2.23, this equation is noticeably different from the arguably more physical K-omega and K-epsilon turbulence model. This is due to the original formulation focusing on convergence, near-wall performance and energy balance. This leads to the model looking more like an empirical relation than a physical representation of the turbulence. Even with this considered, the model performs very well in most aerospace applications although the model does tend to struggle with complex flows such as rotational, free shear and flows with severe separation and so further improvements would be needed to help in these cases (48). Some modifications to model constants have been proposed for swirling flows (49) as well as supersonic flows (50). Overall, the model performs well for its application area with its good accuracy and low computational cost with few drawbacks when applied to the correct flow case.

$$\frac{\partial \tilde{v}}{\partial t} + \vec{u} \cdot \nabla \tilde{v} = C_{b1}(1 - f_{t2})S\tilde{v} + \frac{1}{\sigma} \nabla \cdot [(\nu + \tilde{v})\nabla \tilde{v}] + \frac{C_{b2}}{\sigma} (\nabla \tilde{v}) \cdot (\nabla \tilde{v}) - C_{w1}f_w \left(\frac{\tilde{v}}{d}\right)^2 \quad (2.23)$$

2.3.3 Reynolds stress models

The Reynolds stress models act as a second order closure model for the Reynolds stress tensor. The models work on a shared principal of solving the individual components of the Reynolds stress tensor through individual transport equations. In addition, an extra transport equation works to solve the Turbulent dissipation rate. The Fundamental equations remain the same for each version but the pressure strain correlation Φ_{ij} and turbulent diffusion D_{ij} formulation changes between models like SSG (51) and LRR (52). Reynolds stress models are inherently anisotropic as they solve all components of the Reynolds stress tensor. However, solving them individually with transport equations leads to an additional seven transport equations for these closure models, adding significant computational cost when compared to first order closure models. The two most popular models differ in approach towards the linearity of pressure-strain model with LRR assuming a linear correlation which is not based on reality. A study by Lee and others found that both models handled the pressure-strain correlation well in the mean flow but both struggled in the near wall region due to locality assumptions in the models (53). The general transport

2. Literature review

equation can be seen in equation 2.24 and 2.25. Overall, these models still represent the most physical modelling of the Reynolds stresses using a RANS methodology but are not always suitable for engineering applications due to the computational cost.

$$\frac{\partial \bar{\tau}_R}{\partial t} + \vec{u} \cdot \nabla R_{ij} = \nabla \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_R} \right) \nabla \bar{\tau}_R \right] + P_{ij} + \Phi_{ij} - \epsilon_{ij} \quad (2.24)$$

$$\frac{\partial \epsilon}{\partial t} + \vec{u} \cdot \nabla \epsilon = \nabla \cdot \left[\left(\nu + \frac{\nu_t}{\sigma_\epsilon} \right) \nabla \epsilon \right] + C_{\epsilon 1} \frac{\epsilon}{k} P - C_{\epsilon 2} \frac{\epsilon^2}{k} \quad (2.25)$$

2.4 Anisotropy of turbulent flows

Most modern turbulence models rely on a broad assumption with regards to the nature of the Reynolds stress tensor. The tensor is assumed to be isotropic in most models when this is a fundamentally untrue in nature. Instead, for all wall bounded, rotational and free shear flows, the turbulence is highly anisotropic. To correct for this, turbulence models tend to rely on empirical relations to match the Reynolds stress tensor to experimental or DNS data (54). This assumption works for getting general solutions for lift over a wing or other flow cases but fails to model the detailed turbulence to a physically correct value. This incorrect modelling can cause issues with predictions of: separation, reattachment, wall shear stress, vortex locations and strength and more. Therefore, it is imperative to find modifications to existing models or new outright models which can capture the effects of anisotropy in turbulent flows. This paper aims to explore a novel approach to this with the hybrid anisotropic K-omega SST / stochastic turbulence model.

2.4.1 RANS anisotropic turbulence models

In literature, there is some history of turbulence models which are able to represent the anisotropic Reynolds stresses for turbulent flows. The Reynolds stress models represent one of these groups of models but as covered before are highly expensive to run. There are some examples of anisotropic models using the K-omega SST model as a basis for adaptation. The ASST model (55) introduces non-linear /anisotropic stress strain relationships to the K-omega SST model to enhance near wall modelling of anisotropic Reynolds stresses.

2. Literature review

This model shows promise in particular flow cases however, seems to struggle in simpler flows where these relationships breakdown. The hybrid anisotropic K-omega SST/STM model used in this study (1) (2) has similar foundations with the same base model as ASST. However, the models differ greatly in their formulations with the SST/STM model using a more physical representation of the anisotropic stresses implemented through a stochastic turbulence model rather than direct relations.

In addition to these models, there stands a few novel approaches through completely new turbulence models like EARSIM and Elliptical relaxation models. The EARSIM model (56) attempts to bridge the gap between RSM and eddy viscosity turbulence models by assuming equilibrium conditions for the Reynolds stress transport equations to form brand new explicit algebraic equations. This model works well maintaining frame invariance and realizability however, is limited by higher computational costs than eddy viscosity turbulence models as well as errors from the truncation of higher order terms. The elliptical relaxation models such as the one suggested by Dreeban et al (57) consists of a two transport equation set up similar to other eddy viscosity models. The transport equations model the wall normal stresses and relaxation function to attempt to model the anisotropic Reynolds stress tensor. This model performs well in highly separated flows but can suffer from numerical stiffness near to the wall.

2.4.2 The hybrid K-omega SST Stochastic turbulence model

Hybrid models such as the K-omega SST-Stochastic turbulence model proposed by Könözy (1) (2) combines computational speed of one or two equation models with the anisotropic accuracy of RSM methods. The model works through an adapted Boussinesq equation seen in equation 2.26 where the final term remains unchanged, but the first term in the brackets differs as new stresses are added through the tensor of the anisotropic distribution of the principle turbulent shear stress (1). This new Boussinesq equation allows for the non-diagonal anisotropic terms to be directly calculated through the stochastic turbulence model. The anisotropic terms are weighted by the principle turbulent shear stress seen in equation 2.29. The two transport equations for the anisotropic model can be seen in equations 2.27 and 2.28. Here, the major differences between the two models can be seen with regards to the additional stochastic production term in each equation. This works to add the anisotropic stresses to the energy balance of the equations.

2. Literature review

$$\bar{\tau}_R = \nabla \cdot \{ \rho \mathbf{v}_t [(\nabla \otimes \vec{u}) + (\nabla \otimes \vec{u})^T] + \mu_\Theta \Theta \bar{\bar{G}} \} - \frac{2}{3} \rho \nabla k \quad (2.26)$$

$$\begin{aligned} \rho \frac{\partial k}{\partial t} + \rho (\vec{u} \cdot \nabla) k = & \rho \mathbf{v}_t [(\nabla \otimes \vec{u}) + (\nabla \otimes \vec{u})^T] \cdot \cdot (\nabla \otimes \vec{u}) + \mu_\Theta \Theta [\bar{\bar{G}} \cdot \cdot (\nabla \otimes \vec{u})] \\ & - \rho \beta^* \omega k + \nabla \cdot [(\mu + \sigma_k \mu_t) \nabla k] \end{aligned} \quad (2.27)$$

$$\begin{aligned} \rho \frac{\partial \omega}{\partial t} + \rho (\vec{u} \cdot \nabla) \omega = & \gamma \rho \{ [(\nabla \otimes \vec{u}) + (\nabla \otimes \vec{u})^T] \cdot \cdot (\nabla \otimes \vec{u}) \} + \frac{\gamma}{v_t} \mu_\Theta \Theta [\bar{\bar{G}} \cdot \cdot (\nabla \otimes \vec{u})] \\ & - \rho \beta \omega^2 + \nabla \cdot [(\mu + \sigma_\omega \mu_t) \nabla \omega] + 2\rho(1 - F_1) \sigma_{\omega 2} \frac{1}{\omega} \nabla k \cdot \nabla \omega \end{aligned} \quad (2.28)$$

$$\Theta = \rho a_1 k \quad (2.29)$$

The K-omega SST-Stochastic hybrid turbulence model was validated over three cases by Könözy: for a horizontal smooth cylindrical pipe, for a NACA 0013 Wing and finally for a full jet stream 31 aircraft (58). The experimental data used for validation was from Laufner 1954 (59), Teschner 2013 (60) and Lawson 2017 (61). The studies found good correlation between experimental data and the hybrid model with particular improvements in the circular pipe cases around the Reynolds stress distributions at the outlet of the pipe (58). From the NACA 0013 case, it seems the hybrid model can model near wall vortical structures well in addition to modelling the large vortical structures on the wing to a reasonable degree, although decreasing in ability in high angle of attack cases (58). The paper also reports good correlation between data on the jet stream, k-omega SST and hybrid models. Könözy also notes some areas for future research which there seem to still be gaps in the literature (58). Further work in the tuning of the anisotropic scale factors for high Reynolds number cases is required due to a breakdown in the near wall Reynolds stresses in these conditions has been observed. In addition, Könözy notes that an investigation into using the SST-STM hybrid model as a replacement to the regularly used K-omega SST model for a detached eddy simulation could be an interesting path to investigate (58).

2. Literature review

The hybrid K-omega SST-STM model was validated as part of a study by Mokrane for a case simulating transient flow through a partially filled pipe (62). This study found good correlation for the stationary state simulation. The same model was validated for 2D flow by Könözy (63) over a variety of test cases. The new model was found to match the specific Reynolds stresses more accurately to the experimental data when compared to the K-omega SST model for flat plate and 2D pipe flow benchmark cases. For a 2D wing cases, improvements were again found with modelling the skin friction of the 2D wing (63). Some limitations in the anisotropic flow modelling were found for optimising scale factors for the anisotropic tensor in the various boundary layer structures. Könözy notes that some future work should be performed to produce a blending of the different Reynolds number approaches for the turbulence model to further improve anisotropic flow capturing (63). Some limitations of the hybrid K-omega SST/STM model were also identified in modelling the separation point of flow for a backwards facing step case although the hybrid model seems to capture the reattachment length with less absolute error, 0.003078% compared to 0.11311% (63). However, it is difficult to say with certainty if this is a problem specific to this test case or a wider problem with the model capturing separation without further studies.

Chapter 3

Methodology

3.1 Onera M6 wing case

The Onera M6 transonic wing case is one of the most common validation cases used for turbulence modelling and new CFD approaches. The case is widely used due to its strong experimental results and wide range of supporting literature. This case was proposed for validation by Schmitt et al in 1979 (64), where they presented their experimental data. The experiment covered a variety of flow cases with various mach numbers and angle's of attack. The chosen flow case has an angle of attack of 2.06 degrees and a Mach number of 0.8386. The case consists of an inboard section of the AGARD AR-138 wing, with a rounded wing tip added (65).

The Onera M6 wing validation case described above provides valuable insight into the new models performance in the compressible regime and its ability to deal with supersonic flows. The case is extremely widely used for testing new turbulence models in the presence of shocks and transonic flow and due to it's wide use, there is good documentation of methodologies and some pre-fabricated meshes available for use. For this flow case, a mesh developed by NASA has been used for the initial mesh (66). This was chosen as the mesh has been validated as appropriate and of good quality for analysing this flow case.

The goal of this section is to validate the anisotropic Hybrid K-omega / Stochastic turbulence model for transonic flow conditions in a well tested and used validation case. A mesh study must first be performed, using the anisotropic turbulence model. It was chosen to perform the study using this turbulence model as it is more sensitive to the y^+ of the mesh and so if this model showed convergence on the grid, then the boundary layer should be being captured well. Once the mesh is chosen, a new simulation can be run with

3. Methodology

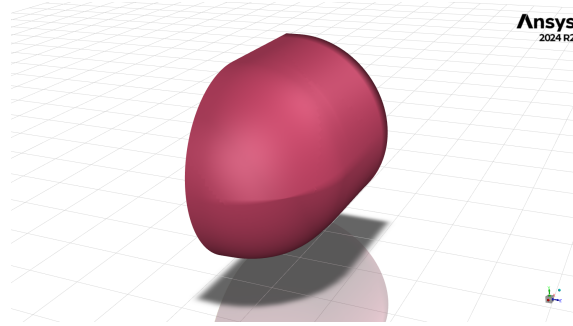


Figure 3.1: Domain for the Onera M6 case

just the K-omega SST model to try determine any difference between the accuracy of flow capturing between the two turbulence models. These simulations may then be compared using the pressure coefficient distribution around the wing for which experimental data is available at seven locations, given as percentages of the wings span: 0.2, 0.44, 0.65, 0.80, 0.90, 0.96, 0.99. In addition, contours can be used to analyse shock wave formation, shear stress and Q criterion iso-surfaces. In addition to comparing the two models, it is also important to analyse the effect of the grid discretisation on results for this case.

3.1.1 Domain

The domain described from NASA, consists of a half bullet shape with a symmetry plane at the root of the wing to reduce computational cost. The remaining boundaries are all prescribed as pressure far fields. By setting these to pressure far-field rather than inlets and outlets, it allows for the angle of attack to be induced via the boundary conditions rather than rotating the mesh or the object inside of the mesh. This is particularly important for this mesh as it has been designed to capture the shocks over the suction surface through further refinement in the region where one could expect their formation. The bullet shaped domain also reduces computational cost when compared to a typical rectangular domain where the upstream corners of the domain have little to no effect on the computational accuracy. The domain can be seen in figure 3.1.

3.1.2 Simulation set up

As described above, the velocity for the simulation can be assigned via the boundary conditions. To do this, the mach number from the experimental data can be inputted into the pressure far-field conditions and then the components in x and y from the set angle of attack produce the correct flow angle. The turbulent properties of the flow can be described

3. Methodology

based on the experimental data's set up. The experimental data comes from a wind tunnel simulation of the Onera M6 wing, therefore hydraulic diameter can be used for the length scale. Hydraulic diameter was calculated from the tunnel height of 2.1m and width of 1.7m to be 1.8789m. In addition, the paper by Schmitt et al. listed the turbulence intensity in the tunnel to be 0.2%.

To match the experimental data, the Mach number and Reynolds number need to be equated to ensure the same flow conditions. Equations for both of these can be seen in equations 3.1 and 3.2. The velocity between both cases is shared and so by rearranging for velocity, a new equation can be formed for temperature seen in equation 3.3. This can then be used to calculate the new dynamic viscosity, see equation 3.4. This temperature of 224.29K and dynamic viscosity of $1.927E - 5Ns/m$ can be used with Sutherland's law to set the fluidic properties of the simulation.

The remaining properties of the air are taken from the experimental data (64) or calculated from ISA tables. Based on the density given by the experimental data, the density is $1.225KG/m^3$, equivalent to sea level conditions, therefore sea level values were used from the ISA table's. The full set of operating conditions can be seen in table 3.1.

$$M = \frac{U_{\infty}}{\sqrt{\gamma RT}} \quad (3.1)$$

$$Re = \frac{\rho U_{\infty} c_{MAC}}{\mu} \quad (3.2)$$

$$T = \left(\frac{Re \mu}{\rho c_{MAC} M} \right)^2 \frac{1}{\gamma R} \quad (3.3)$$

$$\mu_{new} = \frac{\rho c_{MAC} M \sqrt{\gamma RT}}{Re} \quad (3.4)$$

The computational methods chosen were largely decided based on the supersonic na-

3. Methodology

ture of the flow. For the flow, a third order MUSCL scheme was used as literature shows it to perform well in supersonic flow (67) (68). For the gradient discretization, the Green-Gauss Cell based scheme was used for it's strong performance with structured meshing as is in this case (69). All other flow variables were modelled using second order methods. Third order methods were not used for these as they come at a high computational cost. The AUSM Riemann solver was used for capturing discontinuities due to it's better performance with shock discontinuities.

Parameter	Value	Parameter	Value
M_∞	0.8386	Re	1.172×10^7
α [°]	2.06	T [K]	288.15
ρ [kg/m ³]	1.225	S [m ²]	0.7532
b [m]	1.1963	c_{MAC} [m]	0.64607
R [J/(kg · K)]	287	γ	1.4
μ [kg/(m · s)]	1.9269×10^{-5}	ν [m ² /s]	1.57×10^{-5}
I (turb. int.)	0.2%	D_h [m]	1.7599
U_∞ [m/s]	285.3443	α [rad]	0.036
u_x	285.1598	u_y	10.2570
x -flow	0.9994	y -flow	0.0359

Table 3.1: Flow and geometry parameters using engineering notation.

For the turbulence model, the K-Omega SST model was used within the model menu with no options such as curvature correction selected. The additional options were not used as there is no reference of them being tested with the anisotropic model within the literature. Therefore, to ensure a fair test between the two models in their documented forms, it was chosen not to include any additional options. For the anisotropic cases, the same model is used but a user defined function written by Dr László Könözy (2). The set up for this model is described in his literature (2). The model uses its own initialization method as some new flow variables are included in the model. This model also directly computes the Reynolds stress tensor components and so allows for easy plotting of the anisotropic and isotropic Reynolds stresses.

3.1.3 Mesh

Due to the use of a pre-existing mesh, seen in figure 3.2, the decision was taken to refine the mesh using adaptive mesh refinement, a feature inside of fluent. This allows for refinement based on properties of the mesh or flow through criteria outlined by the user or via pre-defined criteria. The first mesh refinement was performed using the pre-defined

3. Methodology

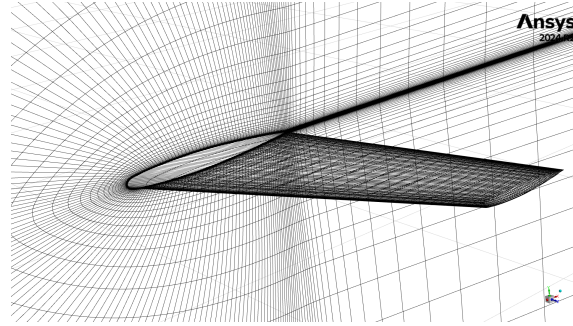


Figure 3.2: Original mesh of the Onera M6 case

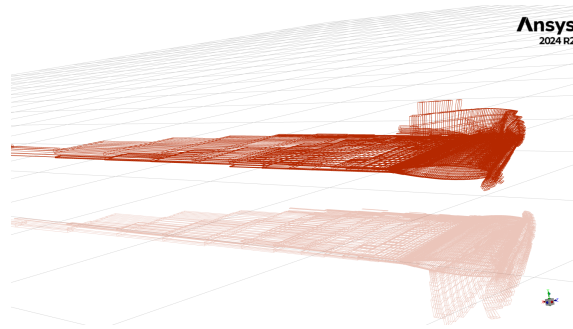


Figure 3.3: Adaptive mesh refinement for density gradient

density gradient based refinement, used for better shock capturing over the suction surface. The second refinement was performed based on the cell volume. This was a user defined criteria, designed to reduce the cell size in the near wall region by limiting the range of cell volume to be similar to the boundary layer cells. An example of how this mesh refinement works can be seen in figure 3.3.

3.1.4 Grid study

The mesh refinement criteria can be seen in table 3.2. The table shows the different criteria for each refinement. The meshes and their parameters can be assessed using a grid convergence study to gauge the effects of further refinement and whether the current refinements are capturing the flow to a good degree.

The grid convergence study for the drag coefficient and for the lift coefficient can be seen in figure 3.4. The figure shows good convergence with each refinement with the third mesh being the closest to the Richardson extrapolated value. The second mesh also seems to converge well compared to the starting mesh.

3. Methodology

Mesh	Coarse	Middle	Fine
Number of cells	294912	630422	2508753
C_L	0.1756	0.1707	0.1671
C_D	0.01145	0.01114	0.01096
Adaption Criteria	-	Density gradient	Cell volume

Table 3.2: ONERA M6 wing mesh study

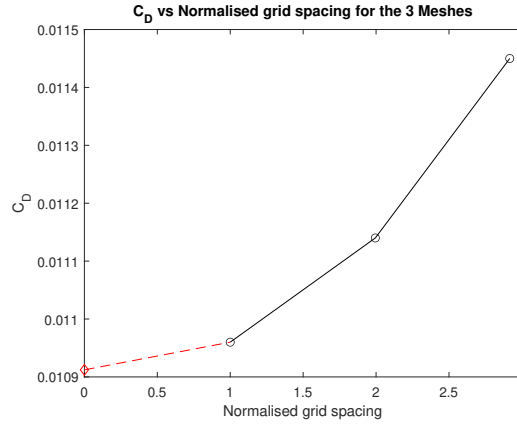


Figure 3.4: Drag coefficient GCI study for the Onera M6 wing case

Similarly to the drag coefficient grid study, seen in figure 3.5, the lift coefficient study shows good convergence when compared to the Richardson extrapolated value. Although it does not come as close to the extrapolated value, there is clear convergence shown in the curve. The finest mesh is closest to the Richardson extrapolated value and the middle mesh is approximately 0.006 off of the extrapolated value.

The second mesh was chosen for the comparison with the K-omega SST case. This mesh was chosen as it has good convergence to the Richardson extrapolated value but also due to some issues with the finest mesh during preliminary studies. These issues came from a seeming oscillatory pressure coefficient distribution over the wing post shock for the anisotropic case. This will be analysed further in the results and discussion section.

3. Methodology

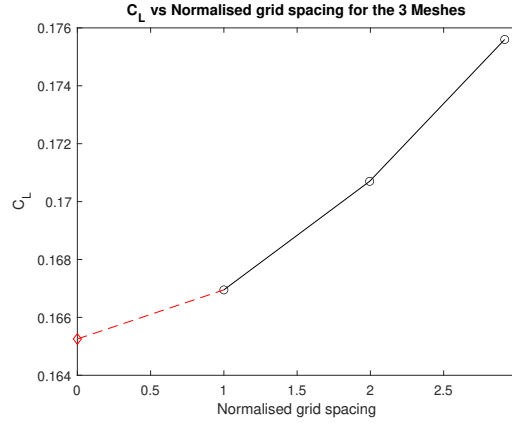


Figure 3.5: Lift coefficient GCI study for the Onera M6 wing case

3.2 Channel flow case

Channel flow cases have been widely used for validation cases of new computational techniques for turbulence modelling. While the Onera M6 case gives insight into the supersonic performance of the model, the channel flow case should provide some insight into the wall modelling capabilities of the model. In order to analyse the wall modelling, it is useful to compare the stress components of the Reynolds Stress tensor $\bar{\tau}_R$. In order to perform this analysis, the components can be taken directly from the user defined memory as a part of the results from the anisotropic code proposed by (1), (2). Therefore, the main complexity is in finding a set of DNS data or experimental data in which the diagonal and anisotropic components are given in the dataset. The DNSlab (70) case was chosen due to its strong documentation and concept as well as the availability of the code to generate the DNS data.

The goal of this study is to measure the ability of the anisotropic hybrid K-omega SST / STM model to capture the Reynolds stresses when compared to DNS data. In addition, the study also allows for some evaluation of the quality of DNS lab as a validation case. The analysis focuses on the Reynolds stress component modelling but also can be used to see how mesh refinement of the near wall region affects the computation and accuracy of the Reynolds stresses.

3. Methodology

3.3 Domain

Two methodologies for recreating the DNS lab were initially evaluated for this project. The first was to recreate the exact case as described in the DNS lab paper. This meant using periodic boundary conditions and a body force to allow the flow to develop. In addition, these simulations would have to be run transiently using URANS, an additional challenge. This in theory was possible however, it was found that to perform these simulations, an additional user defined function would be required for initialisation of data based on the DNS laminar flow in the channel with some fluctuations added to trigger the onset of turbulence. This would add significant complexities in addition to heavy modification of the user defined function developed by Könözy (2). Although possible, this modification went against the principle of validating the turbulence model outlined in his research. In addition, simulating the case using URANS would increase the computational costs significantly for the project, limiting the scope of this project.

Instead, an extended channel methodology was chosen to reproduce the DNS data. This method involved allowing the flow to develop to the desired friction Reynolds number (Re_τ) in a longer channel, using a velocity inlet and pressure outlet to drive the flow. Due to the low speed of the simulation, the pressure outlet was set up to prevent reverse flow and ensure flow was proceeding down the channel. Wall boundary conditions can be applied for the upper and lower walls and symmetry boundaries for the two span wise boundaries.

To calculate the channel length required to reach the friction Reynolds number of 180, DNS data can be used. The DNS data reached the desired friction Reynolds number after 3.67s with a bulk velocity of $5.985ms^{-1}$, therefore the distance travelled along the channel can be calculated to be 22.02m. Adding this to the channel section from the DNS data and multiplying by an additional 1.2m as a safety factor, a channel length for the RANS simulation was calculated to be 34m. The channel height and width should be the same of that of the DNS case at 2m and π m respectively. The domain can be seen in figure 3.6.

3. Methodology

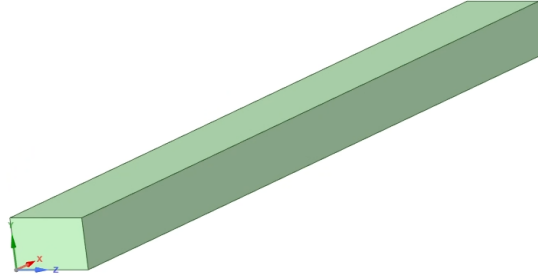


Figure 3.6: Domain used for the channel flow simulations

3.4 Simulation set up

The simulation has been set up as listed previously using a velocity inlet and pressure outlet. The DNS simulation it is based off normalises most aspects using a density of $1\text{Kg}/\text{m}^3$ and so the same density has been applied. Operating pressure and temperature have been set based on this. Velocity was set based on the bulk velocity of the DNS data at a value of 5.985ms^{-1} , due to the channel having no span wise walls. The turbulent length scale was set using a hydraulic diameter of 2m, being the tunnel height. Viscosity was set using the value used for DNS of $1.9E - 3\text{m}^2\text{s}^{-1}$.

Turbulent intensity, for a channel flow has a wide range of possible values and so with no direct listing of a turbulence intensity in the DNSlab paper, a study had to be performed to vary the turbulent intensity on the coarsest mesh. This allowed for selection of a turbulent intensity for the simulation which can be applied to reach the friction Reynolds number of 180. To do this, the x wall shear stress can be plotted along the middle of the walls as shown in figure 3.7. The friction velocity can be calculated for each point along each wall using equation 3.5. The friction velocity along with the half channel height and the kinematic viscosity can be used to calculate the friction Reynolds number at each point along the wall which can then be averaged across both walls to determine where the friction Reynolds number reaches 180. A script can be set up to take the fluent data and find if the friction Reynolds number is reached. Eventually it was found that a turbulent intensity of 1% reached the correct Re_τ . Results from this study can be seen in figure 3.8.

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (3.5)$$

3. Methodology

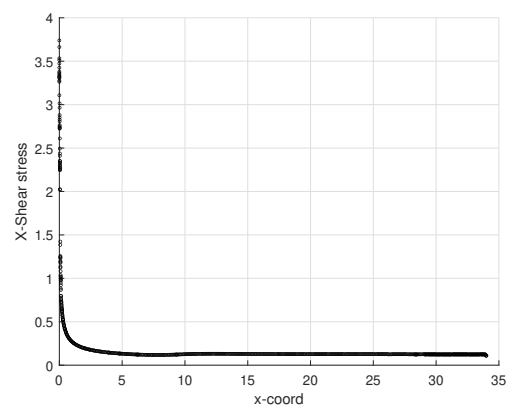


Figure 3.7: X-Shear stress plot along the lower wall of the domain

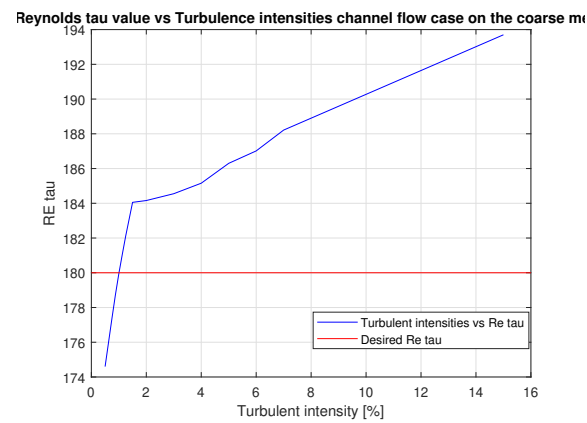


Figure 3.8: Turbulence intensity study

3. Methodology

$$Re_\tau = \frac{u_\tau \delta}{\nu} \quad (3.6)$$

For the computational methods, a pressure based solver is used due to the low Mach number of the flow meaning there should be little to no compressible effects missed using this method. The simulation uses a coupled pressure-velocity solver and least squares based method for the gradient discretisation due to its strong convergence properties. All other flow variables are using second order upwind methods for their computation.

Identically to the Onera M6 validation case, the base turbulent model must be set to the K-omega SST model with no additional corrections applied. The user defined function runs as a compiled function with function hooks to perform the initialisation of the flow. The user defined function outputs as user defined memory all components of the Reynolds stress tensor as raw values. For comparison with the DNS Reynolds stresses, these values must be normalised and made non-dimensional. The method to complete this is listed in Dr Könözy's book (2) and the method varies between the diagonal isotropic and non-dimensional anisotropic components. The method is also listed below in equations: 3.7, 3.8, 3.9, 3.10, 3.11 and 3.12

$$\frac{RMS11}{u_\tau} = \frac{1}{u_\tau} \cdot \sqrt{\frac{|\tau_{11}^{RA}|}{\rho}} \quad (3.7)$$

$$\frac{RMS22}{u_\tau} = \frac{1}{u_\tau} \cdot \sqrt{\frac{|\tau_{22}^{RA}|}{\rho}} \quad (3.8)$$

$$\frac{RMS33}{u_\tau} = \frac{1}{u_\tau} \cdot \sqrt{\frac{|\tau_{33}^{RA}|}{\rho}} \quad (3.9)$$

3. Methodology

$$\frac{\overline{u'_1 u'_2}}{u_\tau^2} = -\frac{1}{u_\tau^2} \cdot \frac{\tau_{12}^{RA}}{\rho} \quad (3.10)$$

$$\frac{\overline{u'_1 u'_3}}{u_\tau^2} = -\frac{1}{u_\tau^2} \cdot \frac{\tau_{13}^{RA}}{\rho} \quad (3.11)$$

$$\frac{\overline{u'_2 u'_3}}{u_\tau^2} = -\frac{1}{u_\tau^2} \cdot \frac{\tau_{23}^{RA}}{\rho} \quad (3.12)$$

3.4.1 Mesh

The initial mesh for this study was a coarse mesh with a desired y^+ of 1. This was desirable as fluent automatically uses wall models at values of over 30 for the y plus. This is important as the aim of this study is to analyse how the turbulence model matches the Reynolds stresses of the DNS data, with the near wall region being crucial to this modelling. The refinement strategy aimed to reduce overall cell size between simulations but with particular focus on reducing y plus. In the literature, Könözsy found that the model captured the anisotropic Reynolds stresses best at a y plus of approximately 0.02 (2) and so that should be the goal for the final mesh. The mesh parameters of the three meshes can be seen in table 3.3.

Mesh	Coarse	Middle	Fine
Number of cells	163047	562792	2093210
RE tau 180 reached	7.7502	7.7413	7.7301
Inflation layers	15	20	40
Growth rate	1.2	1.2	1.1
First layer height (m)	0.00556	0.0032	2.78e-04

Table 3.3: Channel flow mesh study

A grid study was performed for the Channel flow meshes. The three meshes and their

3. Methodology

parameters can be seen in table 3.3. The value used to assess each mesh was the distance along the channel where the friction Reynolds number is first reached for each case. The results of the grid study can be seen in figure 3.9. This shows good convergence in the selected property as the mesh is refined with the best results on the finest mesh but with the second mesh also producing similar values to the Richardson extrapolated value.

3.5 Mesh study

The chosen mesh was the finest mesh for this case. While the second mesh performs well and with a lower computational cost, as stated previously, an extremely low y^+ is required for good prediction of the Reynolds stresses and so it was concluded that the mesh with the lowest y^+ should be used. The mesh quality parameters can be seen in table 3.4. It can be seen that all values are well within good practice limits for CFD modelling. A note should be made of the high average aspect ratio. This is likely larger due to the focus on inflation layer refinement where the layers tend to be extremely thin, depending on the growth rate and first layer height. These high aspect ratio cells can be seen in figure 3.10.

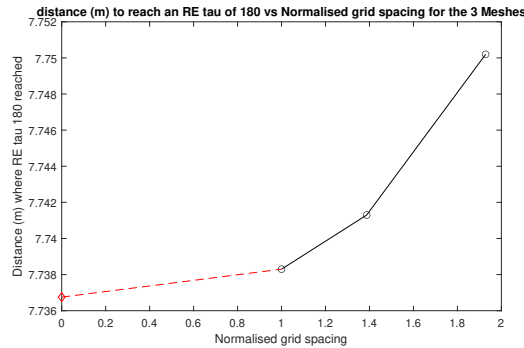


Figure 3.9: Channel flow Grid convergence study

3. Methodology

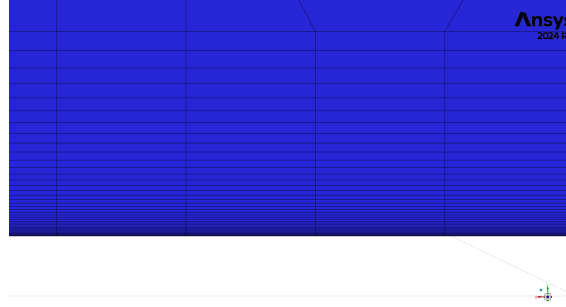


Figure 3.10: Enhanced view of the inflation layers of the finest channel flow mesh

Measure	Value
Average orthogonality	0.98
Average skewness	0.00779
Average aspect ratio	50.20450
Size change	1.57590

Table 3.4: Mesh quality of the finest channel flow mesh.

3.5.1 DNS lab case

The DNS lab code was chosen as the validation for the channel flow case. This paper describes a DNS code which can be run in MATLAB from a series of scripts provided in documentation. The paper by Vuorinen et al. (71) introduces the concept and the results from a 3D channel flow case and a separate 2D case. The channel flow DNS data was validated against an openfoam solver for the same case (72). The code allows for some customisability within its various scripts, including allowing for customisability of the mesh size, some solver settings and the flow initialisation. The code uses a Fourier psuedo-spectral solver instead of a finite element or volume method. This allows for better accuracy in addition to a reduced run time. However, this method is only applicable to relatively simple flow cases. In addition, this code runs at extremely low Reynolds numbers due to the low velocity and density and so the grid resolution does not have to be extremely fine like a typical DNS flow case. This allows for a DNS solver to run on a typical workstation.

The code allows for three separate run modes, each mode describing a different initial flow. Run mode 1 allows for a laminar flow case by fixing the Reynolds number for the simulation. Run mode 2 take the laminar flow profile from run mode 1 and adds random fluctuations to the flow to trigger the onset of strong turbulent flow in the channel.

3. Methodology

Run mode 2 initialises the simulation with the final flow field from run mode 1, meant to improve convergence and stability of the simulation. Run mode 2 was chosen for the validation data. This is to ensure that the flow is as natural as possible without restrictions on Reynolds number or artificial fluctuations added to the flow. This should help with being able to match the DNS case to a RANS flow case. Time step was kept at the default value of $4.5E - 3$, as was the target Courant number of 0.5. The grid was also not modified and so remains at 72 by 72 by 72 cells. These values were kept to the same value used in the paper as the data for this case has been previously validated.

The pressure differentiation scheme uses a second order Runge-Kutta method, a higher order method than the default value was not deemed necessary and would come with a significant increase in computational cost. The "CTanh" variable defines the grid resolution towards the wall. This was kept at it's default value of $5E - 2$. The higher this value the more condensed the mesh becomes at the wall. As stated in the code documentation, this should result in a y plus of approximately 0.6. The temporal solver uses a fourth order explicit method for it's computation.

The DNSlab model calculates when the desired friction Reynolds number has been reached. This is when the average value across the domain on both surfaces has reached the desired value of 180. This may cause some problems with matching the stress tensor components between the DNS lab data and the RANS simulation. This will be discussed further in the results section.

$$\frac{RMS11}{u_\tau} = \frac{1}{u_\tau} \cdot \sqrt{\frac{u'^2}{\rho}} \quad (3.13)$$

$$\frac{RMS22}{u_\tau} = \frac{1}{u_\tau} \cdot \sqrt{\frac{v'^2}{\rho}} \quad (3.14)$$

$$\frac{RMS33}{u_\tau} = \frac{1}{u_\tau} \cdot \sqrt{\frac{w'^2}{\rho}} \quad (3.15)$$

3. Methodology

$$\overline{u'v'} = \overline{uv} - \bar{u} \cdot \bar{v} \quad (3.16)$$

$$\frac{\overline{u'_1 u'_2}}{u_\tau^2} = \frac{1}{u_\tau^2} \cdot \frac{\overline{u_1 u_2} - \bar{u}_1 \cdot \bar{u}_2}{\rho} \quad (3.17)$$

$$\frac{\overline{u'_1 u'_3}}{u_\tau^2} = \frac{1}{u_\tau^2} \cdot \frac{\overline{u_1 u_3} - \bar{u}_1 \cdot \bar{u}_3}{\rho} \quad (3.18)$$

$$\frac{\overline{u'_2 u'_3}}{u_\tau^2} = \frac{1}{u_\tau^2} \cdot \frac{\overline{u_2 u_3} - \bar{u}_2 \cdot \bar{u}_3}{\rho} \quad (3.19)$$

The DNS lab model has some variation in its computation of the Reynolds stress tensor elements. The RANS CFD approach using the anisotropic hybrid model take the raw values before they are converted into non-dimensional RMS values in post processing. It achieves this by calculating each of the elements as user defined memories within the user defined function. The DNS lab case instead solves the flow directly before computing the elements for each grid point by taking the U,V and W and subtracting the mean flow of each velocity component from this to get the u', v', w' values. The fluctuations are then averaged across the span, length, upper and lower surface. Similar to the friction Reynolds number discussed previously, this results in a 1D vector from 1 to 32, giving the fluctuations from the wall to the half channel height. The isotropic and anisotropic elements vary in their computation as in the hybrid anisotropic model methodology. The computation of the isotropic elements can be seen in equations 3.13, 3.14 and 3.15. The isotropic elements are calculated by squaring the fluctuations in each direction to get the equivalent to Reynolds stresses $u'u', v'v', w'w'$. The square root can then be taken and made non-dimensional by the friction velocity. It should be noted that the DNS lab friction velocity has been used for these cases as well as for the fluent data to keep consistency. The anisotropic elements for the DNS lab case are more complicated to calculate, an example of it is given in equation 3.16. This method uses the mean of the cross product of the ve-

3. Methodology

locity components rather than the RMS in its calculation. To acquire the non-dimensional value it is then divided by the square of the friction velocity, similarly to Dr Könözy's approach (2). This can be seen in equations 3.17, 3.18 and 3.19.

The method of averaging across the domain is not suitable for the RANS approach for a variety of reasons although it may improve the accuracy. This is due to the mesh sizes, while the DNS case has fewer cells than the RANS simulation, meaning to export all the data and process it would take a much longer time. In addition, the question could be raised of what region to average the cells for the RANS simulation, whereas the DNS simulation has a periodic boundary and so the region to be averaged is pre-defined. For the fluent case, an arbitrary distance up and downstream would have to be chosen. In addition, it would be impractical to pull the data from fluent as it would likely require extracting at planes placed a finite distance apart, leading to a jagged distribution of Reynolds stresses. Another option is exporting and sorting the whole data set from fluent however, this is impractical and computationally expensive.

Chapter 4

Results and Discussion

4.1 Onera M6 validation case

As discussed previously, the Onera M6 wing validation case provides a valuable validation case for transonic flow conditions for the anisotropic turbulence model. In this case although the flow is below a Mach number of one, will see shocks on the suction surface of the wing where the local Mach number can exceed one. In addition, the lower surface can experience these localised supersonic regions but likely only on the wing tip. As well as testing the physical modelling with the use of adaptive mesh refinement, it is interesting to note how the model will perform when using some features of commercial CFD solvers.

4.1.1 Shear stress and Shock capturing

The supersonic regions of the upper (suction) surface of the wing have been displayed through banded contours where the Mach number is greater than one at each of the seven locations explored in the NASA experimental data (64). The distribution over the wing for each of the three meshes and for the case on the second mesh with the K-omega SST model can be seen in figure 4.1.

Looking first at the different meshes in figure 4.1 (a),(b) there are some noticeable differences between all three of the meshes. For example, the first mesh seems to have a slight secondary shock at the 96% span location. This is likely a grid-induced anomaly as this is present on the coarsest mesh. In addition, the middle mesh seems to capture the shocks at the 80% span location more accurately than the coarser mesh, shown by the more definitive boundary between the two shocks before they merge at a further span wise position. This is also present in the 65% span wise location where there is a clearer gap between

4. Results and Discussion

the two shocks on the middle mesh than the coarse one.

Comparing the middle mesh to the finest mesh shows some surprising results. As mentioned previously, there were some doubts in the suitability of the third mesh. This primarily came from convergence issues with the mesh, meaning it took much longer to converge the lift and drag coefficients when compare to the previous cases. The direct cause of this is difficult to say for certain however, some possible reasons will be discussed here and later in the report. The major difference between the meshes seen in figure 4.1 (c),(d) are with regards to some small isolated shocks downstream of the initial shocks on most of the outboard locations on the wing for the finest mesh. This is most likely a grid related error with regards to the adaptive mesh volume refine. The smallest cells were refined to try and reduce the y plus of the grid. However, this may have lead to issues with the mesh becoming non-conformal. For example, it could be that a cell is refined using the adaptive mesh refinement, but the cell up or downwind is not refined, leading to hanging nodes on the larger cells face. This can cause instability, possibly causing the spurious shocks seen downstream. In addition, large changes in cell volume may result in varying growth ratios of the boundary layer cells, affecting the boundary layer capturing. In addition, this may be caused by Riemann solver settings such as the limiter of the third order MUSCL scheme used here. This, combined with a slightly distorted grid, could be causing the small shocks. The issue may be a combination of all listed above, or just one. Further analysis of contours and pressure coefficient may help to determine the nature of this issue.

Finally, analysing the differences in the two turbulence models on the same mesh in figure 4.1 (b),(d), the results seem to be remarkably similar. Both models predict the shocks with similar shape and structure, with the only noticeable difference being in the modelling of the 65% span location. At this location, there is a slight change in the profile of the secondary shock after the small expansion post front shock. This variation is tiny, with the main difference being the depth but also slight profile changes. However, overall the models match each other very well for this case. This helps to validate the hybrid K-Omega SST/STM model as it compares well to an already established and widely used turbulence model.

4. Results and Discussion

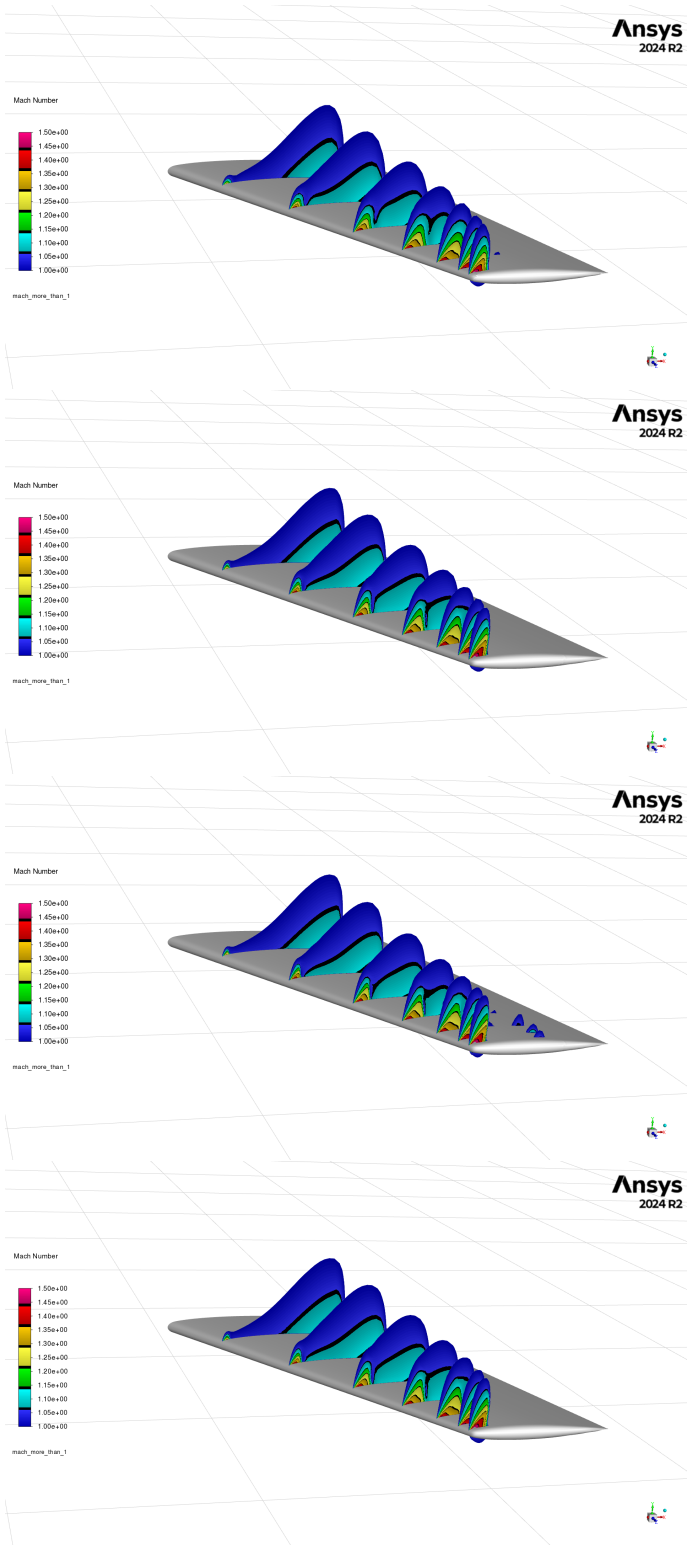


Figure 4.1:
(a) Mach number contours for mesh 1 Anisotropic model
(b) Mach number contours for mesh 2 Anisotropic model
(c) Mach number contours for mesh 3 Anisotropic model
(d) Mach number contours for mesh 2 K-omega SST model

4. Results and Discussion

The shocks provide some detail in the ability of the turbulence model to capture the transonic flow however, downstream modelling of post-shock flow is also vital for any supersonic / transonic flow case. Analysing this can also help to see how the turbulence model behaves through discontinuity and if the applied Riemann solver is working correctly in conjunction with the anisotropic model. To analyse this, the shear stress contour have been plotted for the Onera M6 wing case for all three meshes and for the K-Omega SST model on mesh 2. This can be seen in figure 4.2.

As mentioned above, the shear stress on the wing can be used to analyse what is happening post shock to the boundary layer. High wall shear stress is associated with attached and largely laminar flow. A reduction in the shear stress usually corresponds to a transition from laminar to turbulent flow. Shocks play a large role in the shear stress distribution over the wing. The change in flow conditions across a shock boundary tend to disturb the laminar flow, inducing a transition to turbulent flow post shock. The shock triggers the transition due to the sudden pressure rise, resulting in a strong adverse pressure gradient. In addition, a general pattern of lower shear stress at the tip may be observed due to some separation effects due to the 3D flow effects caused by the wing sweep. Included in these 3D effects is the stronger shock at the tip but this is more with regards to the reduction in chord length at the wing. This results in a stronger favourable pressure gradient about the leading edge to the maximum thickness point of the wing leading to a greater acceleration of flow in this area. This can be observed in figure 4.2.

Comparing the first and second meshes in figure 4.2 (a),(b), both seem to model the expected shear stressed on the wing well, with the contour produced being similar to what is expected with regards to the shock and wing tip vortex for the wall shear stress. There are some subtleties of which the middle mesh seems to capture better. For example, the interaction between the wing tip vortex and the post shock flow seems to have a more distinct separated region of low wall shear stress when compared to the coarse mesh. The post shock low wall shear stress region for the middle mesh seems to be more uniform and smooth in its distribution when compared to the coarsest mesh. However, with all of this taken into consideration, both meshes still perform well.

4. Results and Discussion

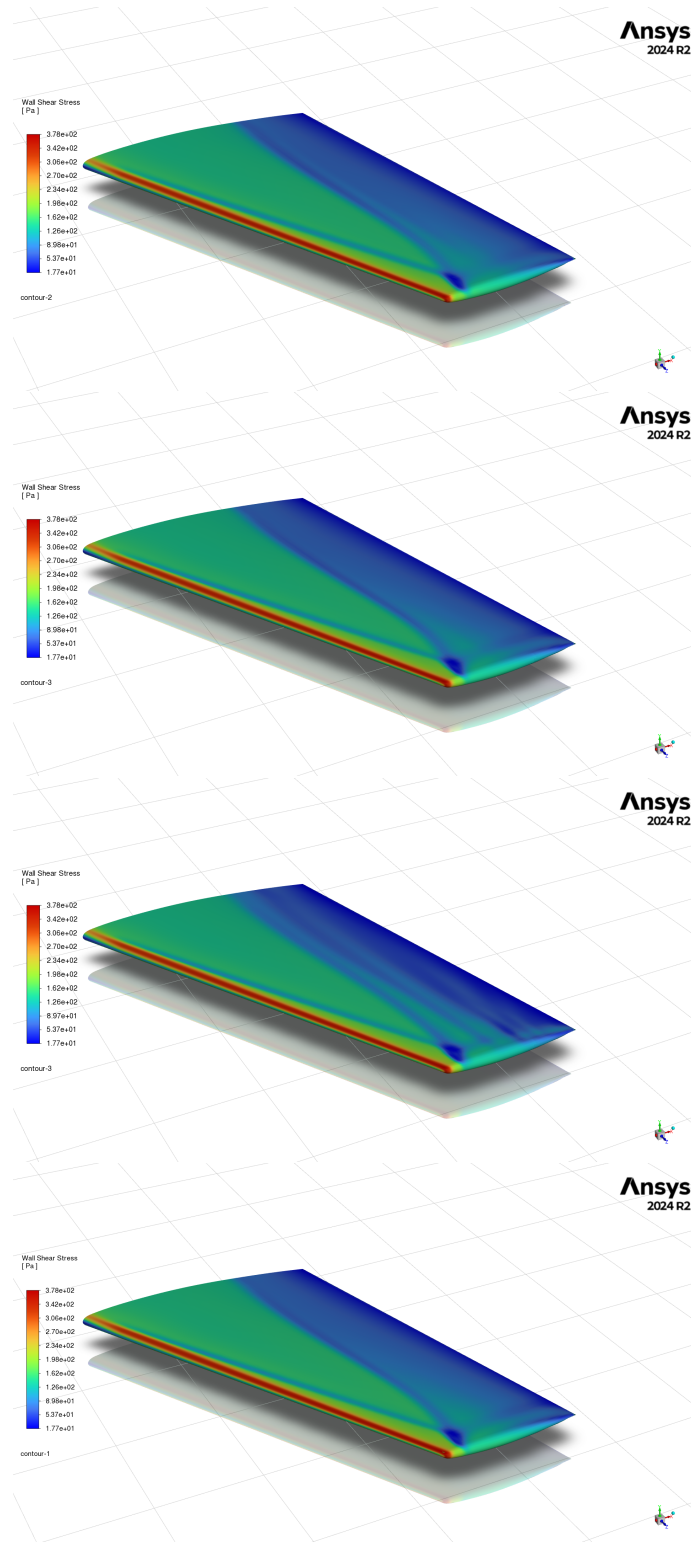


Figure 4.2:

- (a) Shear stress contours for mesh 1 Anisotropic model
- (b) Shear stress contours for mesh 2 Anisotropic model
- (c) Shear stress contours for mesh 3 Anisotropic model
- (d) Shear stress contours for mesh 2 K-omega SST model

4. Results and Discussion

Similarly to the shock distribution over the suction surface of the airfoil, the fine mesh also seems to struggle with modelling the wall shear stress. This can be seen when comparing figure 4.2 (b),(c). Here it can be seen that while the middle mesh has two distinct drops in wall shear stress, characterised by a shock at these locations, the finest mesh has six distinct lines, best viewed at the wing tip. These shear stress drops are small however, still indicate the incorrect modelling of additional smaller shocks along the wings surfaces. The modelling of the main lambda shock and leading edge regions seems to remain strong for this case. It is largely the downstream region which is significantly effected. This is likely linked to the previous refinement working mostly in the shock region, with the parameter used for refinement being the density gradient. Therefore, this region is well refined already, but the region behind may then suffer from some inconsistency in cell sizing due to the volume refine conditions. This can lead to some post shock dispersion and localised boundary layer thickening at the mesh. This is most likely what is causing the small shocks in the downstream flow rather than an error caused by the Riemann solver as these usually perform well when tested for commercial solvers such as Fluent.

Comparing the two turbulence models, both models seem to model the problem to a very good degree with all structures existing as expected. This is shown in figure 4.2 (b),(d) where both models produce similar results for the lambda shock region as well as in the upstream conditions. Some difference can be observed in the wing tip shear stresses. For example, in the separated region behind the wing tip vortex and shock interaction, the anisotropic model seems to produce a lower downstream shear stress indicating a stronger interaction. This is consolidated by the fact that the upstream point where the two structure meet seems to have a larger reduced shear stress region. This difference observed comes from the increased ability of the anisotropic model to represent the wing tip vortex which itself is highly anisotropic in nature. This same enhancement in vortex modelling from the anisotropic model can be seen when analysing the normalised Q-criterion contour in figure 4.3 (b),(d) and is further enhanced through similar findings in the capability of the model in capturing vortical structures on a NACA 0013 wing in Dr Könözy's work (2). Finally, the turbulent post shock flow in the K-Omega SST case seems to be smoother in nature than that of the anisotropic model. This is unlikely to be a factor of the grid refinement and more likely to be related to the K-Omega models characteristic of modelling weaker shock signatures. This was also shown by Raje et al. when comparing a K-omega SST model to an anisotropic turbulence model developed by the authors (73).

4. Results and Discussion

4.1.2 Turbulence capturing

Iso-surfaces in CFD are a useful tool in analysing regions in which flow properties are of a single or a range of values. This may be used in conjunction with the normalised Q-criterion values over the domain to produce surfaces showing the vortical structures. By using the normalised Q-criterion value of zero, an analysis of the vortical structures and shear strain dominated regions can be derived. These can be seen in figure 4.3 where the normalised Q-criterion iso-surface is displayed with contours of the velocity magnitude plotted onto it.

Analysing the differences between the coarsest and middle mesh in figure 4.3 (a),(b), a clear difference in the vortical structure can be observed. For the middle mesh, the vortical structure can be seen with more detail. This is especially present in the interaction of the vortex with the sharp outboard shock. The detail in the interaction on the wing itself can be observed where the decrease in shear stress was also observed in figure 4.2 (b). In addition, the wake caused by the separation can be seen by the thicker shear layer behind the wing at this location. The shear layer in general is seen to be thicker on the medium mesh when compared to the coarser mesh.

As discussed with the previous contours, the finest mesh seems to struggle with the post shock region with spurious shocks seen in the downstream section of the wing. This can be seen again in 4.3 (b),(c). For mesh two, the shock boundaries can be observed through the breaks in the iso-surface over the wing. Similarly, the finest mesh shows these shocks but also the additional shocks through more gaps in the iso-surface. In this contour, the downstream effects can be observed as a large increase in the thickness and size of the shear layer behind the wing caused by the increase in turbulence and likely separation due to the small additional shocks. The extra shocks can also be seen to interact with the wing tip vortex making the vortical structure downstream smaller and less defined. The reduction in strength of the vortex comes from the decrease in velocity across the suction surface as seen in the contour near the wing tip. This results in an increase in pressure and so a reduction in the pressure differential between the surfaces, reducing the ability of the air to swirl about the tip due to a weaker favourable pressure gradient about it.

4. Results and Discussion

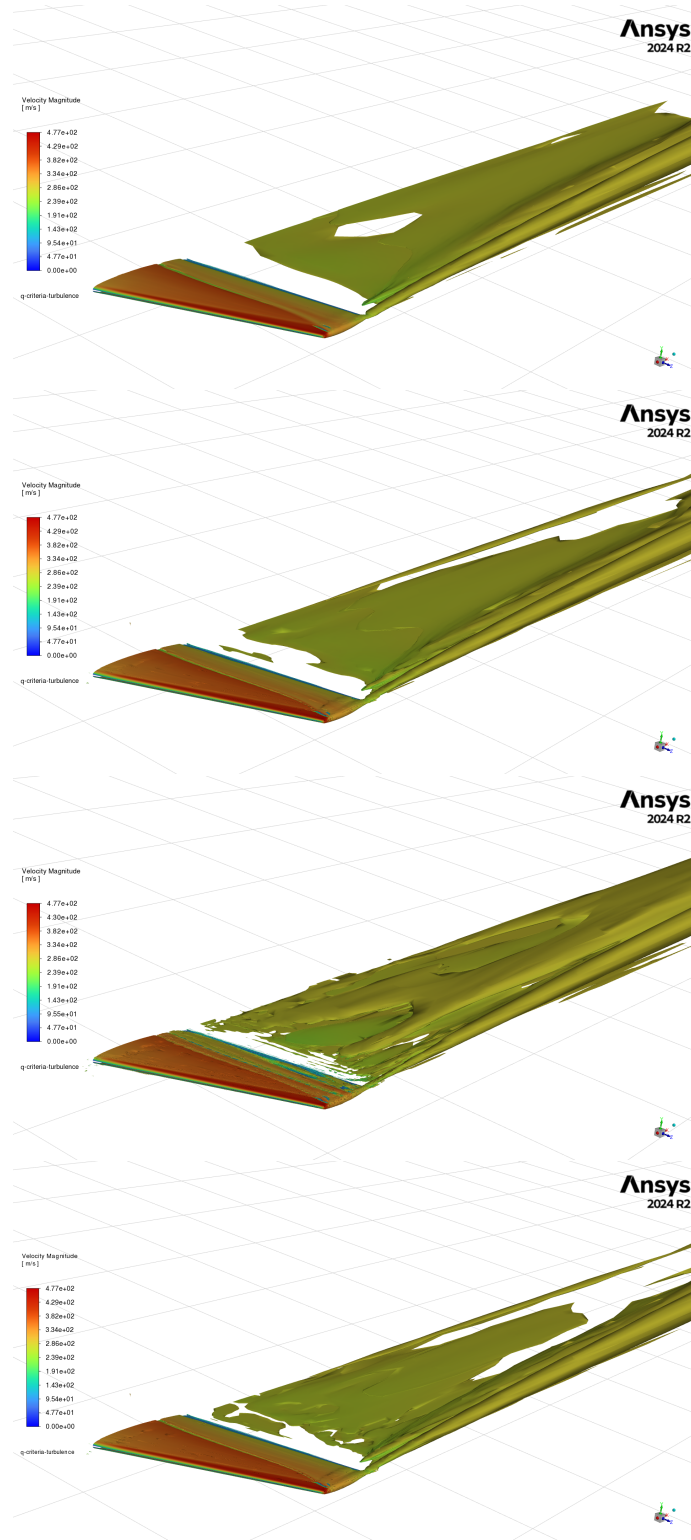


Figure 4.3:

- (a) Velocity contours on Q-criterion iso-surface for mesh 1 Anisotropic model
- (b) Velocity contours on Q-criterion iso-surface for mesh 2 Anisotropic model
- (c) Velocity contours on Q-criterion iso-surface for mesh 3 Anisotropic model
- (d) Velocity contours on Q-criterion iso-surface for mesh 2 K-omega SST model

4. Results and Discussion

Finally, looking at the difference between the two turbulence models in figure 4.3 (b),(d), there are some clear differences in the two models, particularly with reference to the shear layer behind the wing. The shear layer seems to propagate further downstream for the anisotropic model likely due to the anisotropic model's reduced damping post shock when compared to the K-omega SST models purely isotropic eddy viscosity dampening. In addition to the shear layer differences, there is a clear differences in the vortical structures at the wing tip for both cases. The wing tip vortex for the K-omega SST model seems to be less tight and has some small secondary vortices breaking off downstream. The tip shock-wingtip vortex interaction seems also to be dampened for the K-omega SST case when compared with that of the hybrid K-omega SST/STM model.

4.1.3 Pressure coefficients and absolute error

4.1.3.1 Mesh analysis

In order to validate the computational results, some comparison must be made not just between models and mesh performance but also with experimental data. The paper by Schmitt et al (64) measures the pressure coefficient at seven separate locations described in the methodology section of this report. These pressure coefficient distributions about the wing at each location are plotted below against the two turbulence models and also against the three meshes to once again justify the choice of mesh but also the effects of adaptive mesh refinement. In addition to the qualitative analysis, to fully validate the model, quantitative analysis of the absolute errors have been presented in the form of norm 1, norm 2 and norm max.

The pressure coefficients for each mesh can be seen in figure 4.4. First looking at the starting mesh, there is some good correlation between the computational and experimental data for all cases. The main source of difference between the two seems to come with the post shock pressure on the upper surfaces. In all cases, the computational data seems to overshoot the pressure recovery post-shock. In general, the pressure coefficient on the coarsest mesh seems to then decrease again to match the experimental data. This is largely the case for the locations 1-3 however, after this outboard position, when the shocks begin to become sharper and closer to oblique, the mesh struggles to re-align with the experimental data. Instead, the pressure coefficient of the coarse mesh tends to oscillate further before reaching the C_p of the experimental data. These oscillations seem to get progressively longer as the location moves further outboard. This is due to the grid

4. Results and Discussion

spacing being too large to reliably dampen out large oscillations in flow values even with a third order numerical scheme like MUSCL being used.

Comparing the coarse mesh with the middle mesh, in figure 4.4, it is clear again why this mesh was chosen for the further studies of comparing turbulence models. The middle mesh shows extremely good correlation with experimental data at most locations when compared to the middle mesh. This is characterised by its much better behaviour when modelling the pressure rise after the shocks. The middle mesh does not overshoot the pressure rise for any location. However, there does seem to be some inconsistencies between the pressure coefficients on the middle mesh and the experimental data. In particular, the C_p values within the lambda shock structure seem to be misrepresented, which can be seen in locations 1-5. Here, in general, the initial pressure increase after the first shock structure tend to over estimate the size of the shock by a small amount and seem to model the secondary shock earlier than expected. As the shocks converge towards a single sharper shock for locations 6 and 7, the pressure coefficient is represented very accurately on the mesh. Both meshes also tend to see the pressure differential reduce to near zero post shock when in fact for the further inboard locations, this isn't necessarily true as per the experimental data. Overall though, the middle mesh shows significant improvements over the coarsest mesh.

Finally, looking at the pressure distributions for the finest mesh in figure 4.4, once again it can be seen why this mesh was avoided for further study. The final mesh seems to represent the lambda shock region to a good degree, if not a small amount worse than the middle mesh but does improve upon the coarse mesh in general. However, as discussed with regards to figures 4.1 (d) and 4.2 (d), it is in the post shock region where this mesh suffers from small spurious shocks in a region where these would not be expected. This is shown clearly in each locations pressure coefficient plot. Shown most clearly as the shocks become sharper and closer to oblique, there are significant oscillations in the pressure coefficients which do not correspond to any fluctuations in the experimental data. These are caused by the sharp fall in pressure pre-shock and subsequent rise in pressure post shock. This again leads to the conclusion that these oscillations are not caused by an instability in the turbulence model or computational method as the period of the oscillations are not consistent, nor are the amplitudes, indicating that these come from additional shocks and not a lack of physical damping capabilities of the model. Instead, these are issues encountered through the inconsistent refinement from the volume refine leading to localised boundary layer thickening and post-shock dispersion.

4. Results and Discussion

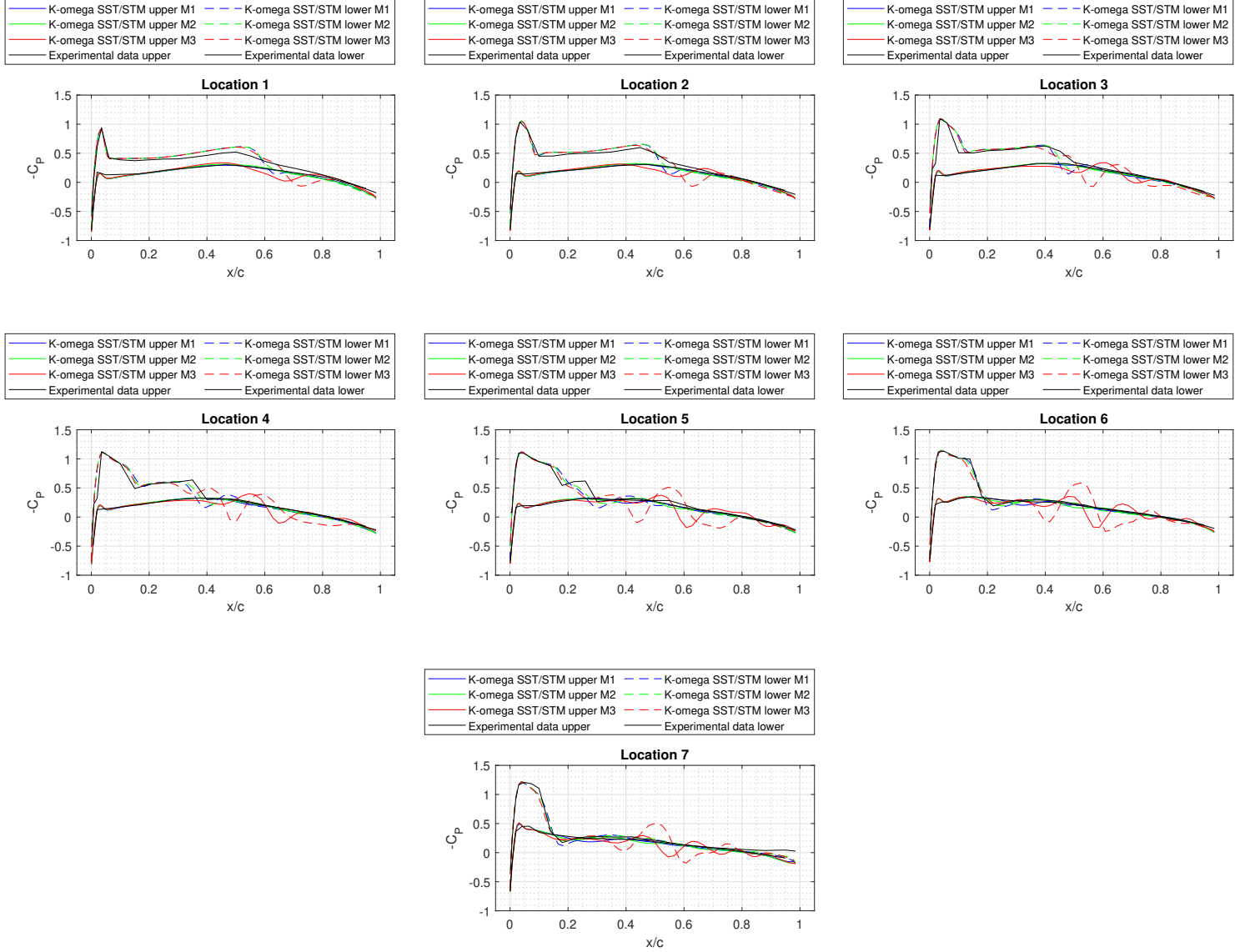


Figure 4.4: Comparison of mesh performance at capturing the pressure coefficient compared to experimental data

4. Results and Discussion

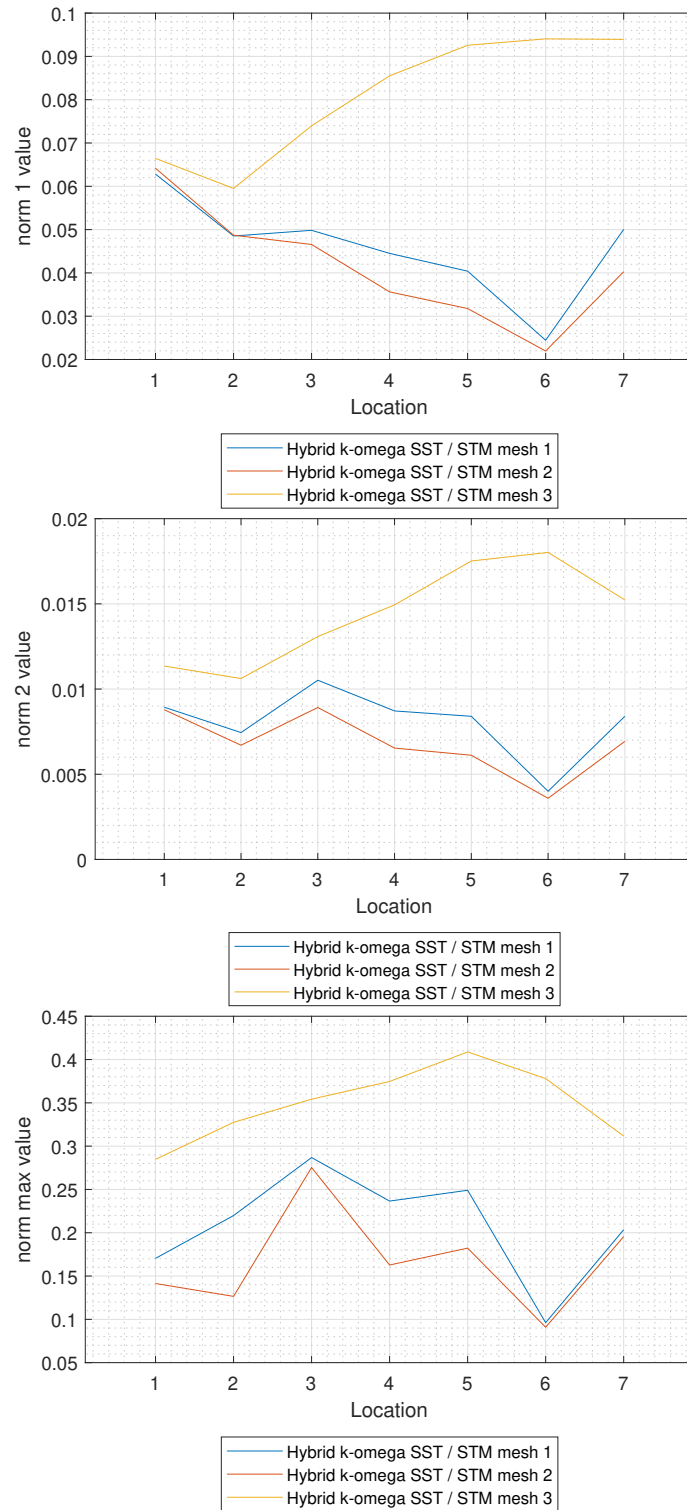


Figure 4.5:

- (a) Norm 1 comparison of errors from the three meshes
- (b) Norm 2 comparison of errors from the three meshes
- (c) Norm max comparison of errors from the three meshes

4. Results and Discussion

The norms seen in figure 4.5 (a) are used to quantify the errors identified in the previous section. The norm 1 values are calculated by taking the sum of absolute errors about both surfaces of the wing at each location and dividing by the number of sample points. This value helps to ascertain the average error by weighting each absolute error evenly. Immediately, it is clear that the third mesh and its spurious shocks post main shock shows the highest norm 1 values. The gap between the finest mesh and the remaining two meshes only grows as the location moves further outboard. It is clear that all three meshes struggle to keep absolute error low for the furthest outboard section due to the complex interaction between the wing tip vortex and the sharp shock at this location. Comparing the middle and coarse mesh, both perform similarly with regards to norm 1 for the first two locations however, looking further outboard, the middle mesh significantly out-performs the first mesh. Looking again at figure 4.4 (a), this difference seems to largely be coming from the coarse mesh's tendency to overshoot the pressure rise post shock. At the furthest outboard section, the two meshes both have a sharp increase in their norm 1 value. Some of this error seems to come from the modelling of the pressure recovery post shock but in addition, the experimental data shows a large pressure difference between the trailing edges as the upper surface is likely stalled and so has a recirculating region where the pressure is higher on the suction side. However, neither mesh seems to capture this and so final pressure coefficients converge.

The norm 2 values in figure 4.5 (b) are calculated based on the summation of the squared absolute errors which are subsequently square rooted. Calculating the norm this way puts additional weighting onto larger errors produced. Looking at the norm 2 values of the finest mesh, again the error produced by this grid is significantly higher than the remaining two meshes. However, the norm 2 value does seem to drop for the furthest outboard section indicating that it is largely smaller errors in the pressure coefficients which are producing such a high norm 1 value. For the coarse and middle mesh, it can be noted that for the norm 2 value at the inboard locations, the middle mesh is now lower than the coarse mesh, indicating that its errors are smaller but more numerous than in the coarse mesh for the first two locations. This can be seen in figure 4.6 where the coarser mesh overshoots the pressure increase, producing a large error whereas the middle mesh seems to have a smaller error at this point but then more error at the trailing edge. There is an increase in norm 2 value at the third location for both meshes where it seems the errors are larger in nature but less common. This was not clear in figure 4.5 (a) but is from both meshes predicting the initial shock later than the experimental data. As the locations go further outboard, the norm 1 and norm 2 distributions seem similar in profile and so it can be concluded that the errors were of similar magnitude at these locations.

4. Results and Discussion

The norm max values in figure 4.5 (c) show the maximum absolute error at each location for the mesh. It can be seen that for the coarsest mesh, the maximum errors produced increase as the location moves outboard however, there is a noticeable decrease towards the furthest outboard section. This corresponds to a reducing amplitude of the spurious oscillations. For the coarse and middle mesh, the largest errors arise at the third location as discussed in the norm 2 section and tend to reduce again moving outboard until the furthest outboard section. This further validates the findings of the norm 2 section. Still for this case and for all other norms, the middle mesh performs significantly better.

4.1.3.2 Turbulence model comparisons

The two turbulence models K-Omega SST and the hybrid anisotropic K-Omega SST/STM model, can be compared to the experimental data with regards to pressure coefficient in order to validate their use for transonic flows. This is shown in figure 4.6 where it is immediately clear that both models perform extremely well and with good alignment to the experimental data. Looking at location 1, both models perform identically for the modelling of the initial shock and match the experimental data well. Moving further along the chord, both models seem to over predict the pressure drop through the lambda shock with both lines being noticeably above the experimental data. This leads to each of the models predicting a later secondary shock than the experimental data. Some difference between the two models can be observed on the pressure surface. Initially, both models seem to suffer a small fluctuation in pressure after the initial pressure drop about the leading edge of the wing. The pressure then recovers in line with the experimental data at which point both turbulence models begin to under predict the pressure of the lower wing surface although the anisotropic model does seem to model this better. This is likely due to the enhanced representation of the transitioned turbulent flow at this point on the wing by modelling the anisotropic Reynolds stress tensor elements in the anisotropic model.

A similar trend to location 1 is observed for location 2 in figure 4.6 where the main source of difference between the experimental data and turbulence model comes from the secondary shock, although both models perform similarly with regards to this. Again the anisotropic model is representing the pressure distribution better than the K-omega SST model on the lower wing surface. Location 3 however, shows some more differences in the behaviour of each turbulence mode. This is best observed at the secondary shock where the anisotropic model sees a small improvement in shock location prediction compared to the experimental data although it seems to over predict the pressure gradient across the shock resulting in a small overshoot in pressure recovery across the shock.

4. Results and Discussion

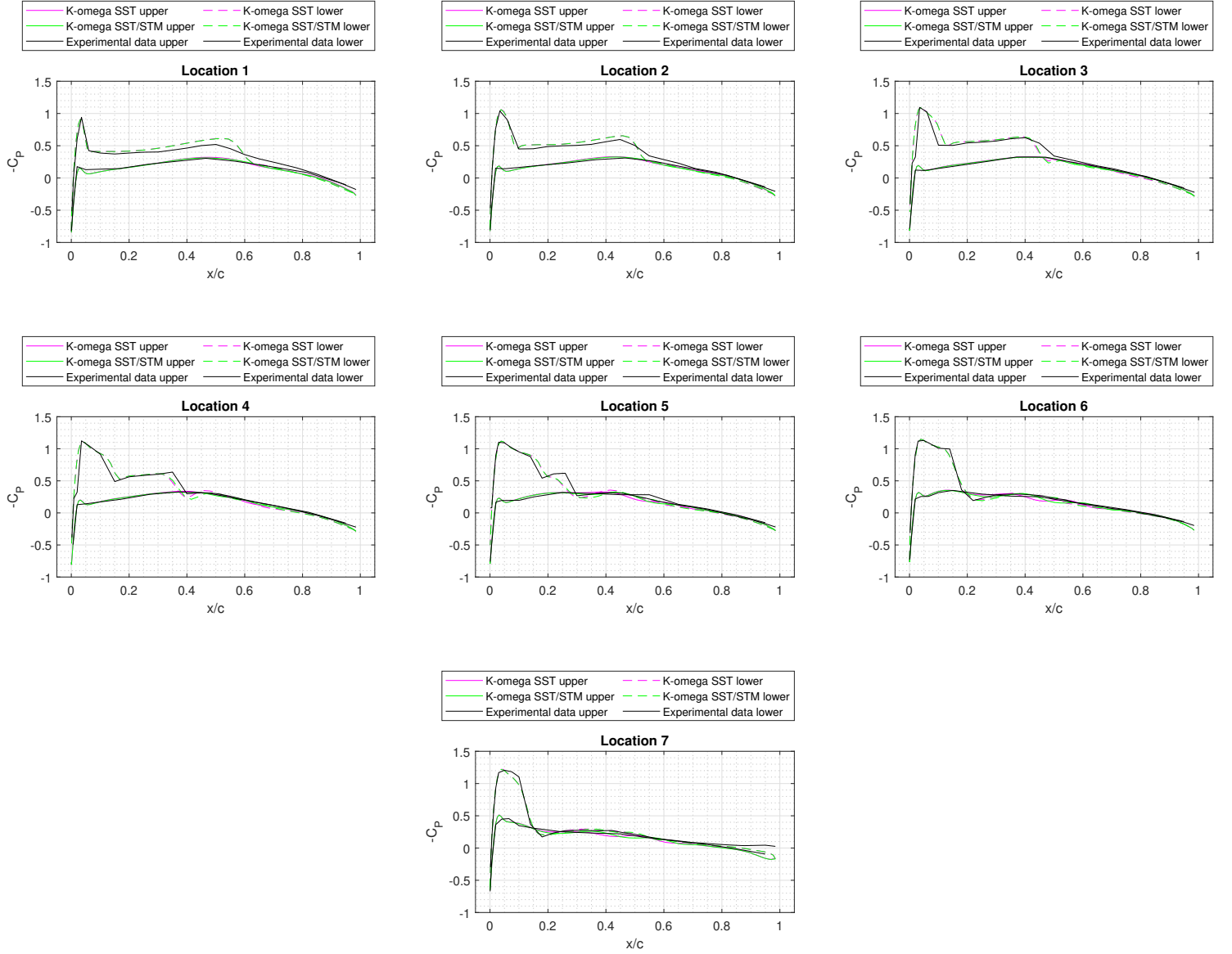


Figure 4.6: Comparison of turbulence model performance at capturing the pressure coefficient compared to experimental data

4. Results and Discussion

This is likely caused by the anisotropic models reduced isotropic dissipation reducing the numerical dampening post-shock. This small drawback can be further improved by further calibration of the model constants. After this discrepancy, the pressure coefficients of the anisotropic model recover to the experimental values and even improve upon model accuracy when compared compared to the K-Omega SST model when approaching the trailing edge of the wing.

Locations 4, 5 and 6 show the same patterns as those of location 3, as seen in figure 4.6 with a slight increase in the overshoot post shock by the anisotropic model. Similarly to location 3, location 4 also struggles to capture the nature of the first shock for both turbulence models. Again for these locations, the modelling of pressure coefficients post shock and approaching the trailing edge is modelled much better with the anisotropic model than the K-Omega SST model. Interestingly at locations 5, 6 and 7, there seems to be some oscillations in the pressure coefficients after the lambda shock region. This seems to coincide with the anisotropic model not tending to overshoot the pressure recovery post-secondary shock. In addition, the secondary shock for location 5 is modelled poorly by both of the turbulence models. The cause of these oscillations could be attributed to an under-damping of both models from the eddy viscosity. In addition, factors like the numerical scheme should be considered, perhaps a higher order method for all flow variables may be used to improve the gradient capturing of the inter-cell fluxes. This instability may be caused by the two shocks present being close in locations 5 and 6 as the oscillations seem to be less severe at location 7 where the two shocks seem to merge into a sharp singular shock. At location 4 but also for locations 5 and 6, a more distinct pressure difference between the upper and lower surface towards the trailing edge can be observed indicating a better resolution in the turbulence on both surfaces of the wing. This is particularly prevalent when comparing the trailing edge of the anisotropic model where there is clear separation between the surfaces even if there is some error in the final pressures observed.

These observations of the physical model are useful for analysing the short comings of each model but to gain a full picture of the models performance, a look into the absolute error norms is crucial. The norm 1 values for the K-Omega SST and the hybrid anisotropic K-Omega SST/STM model can be seen in figure 4.7 (a). The overall trend in these results is that the average error of both models at each location is extremely low with the highest errors observed at the first location with values of 0.064 for the anisotropic model and 0.061 for the K-omega SST hybrid model at location 1. This higher error seems to come from the misrepresentation of the secondary shock by both turbulence models.

4. Results and Discussion

The difference between the turbulence models is likely coming from the K-Omega SST model representing the trailing edge to a greater degree for this case. The next largest difference in errors seems to be at locations 4 and 5 where in both cases the anisotropic model outperforms the K-Omega SST model by a reduction in error of approximately 0.002 for both locations. As noted previously, the further outboard the location, the better the anisotropic model is at modelling the mid chord to trailing edge pressures. With little other evidence of discrepancies between the models it seems as if this is the point of difference between them, The norm 1 value continues to reduce as the locations become further outboard on the wing before increasing at the tip. It is interesting to consider that both turbulence models tend to follow the same pattern in which absolute error increases and decreases by similar amounts and at the same locations on the wing. This may be related to meshes being more sensitive at certain flow conditions on the wing meaning any turbulence model would struggle more at these conditions however, it may also be from the fact that the methodologies behind each of these turbulence models are similar with the anisotropic model being an extension of the base K-Omega SST model. Therefore, this pattern of errors is likely from the numerical formulation of the base K-Omega SST model rather than an issue with the mesh at these locations.

4. Results and Discussion

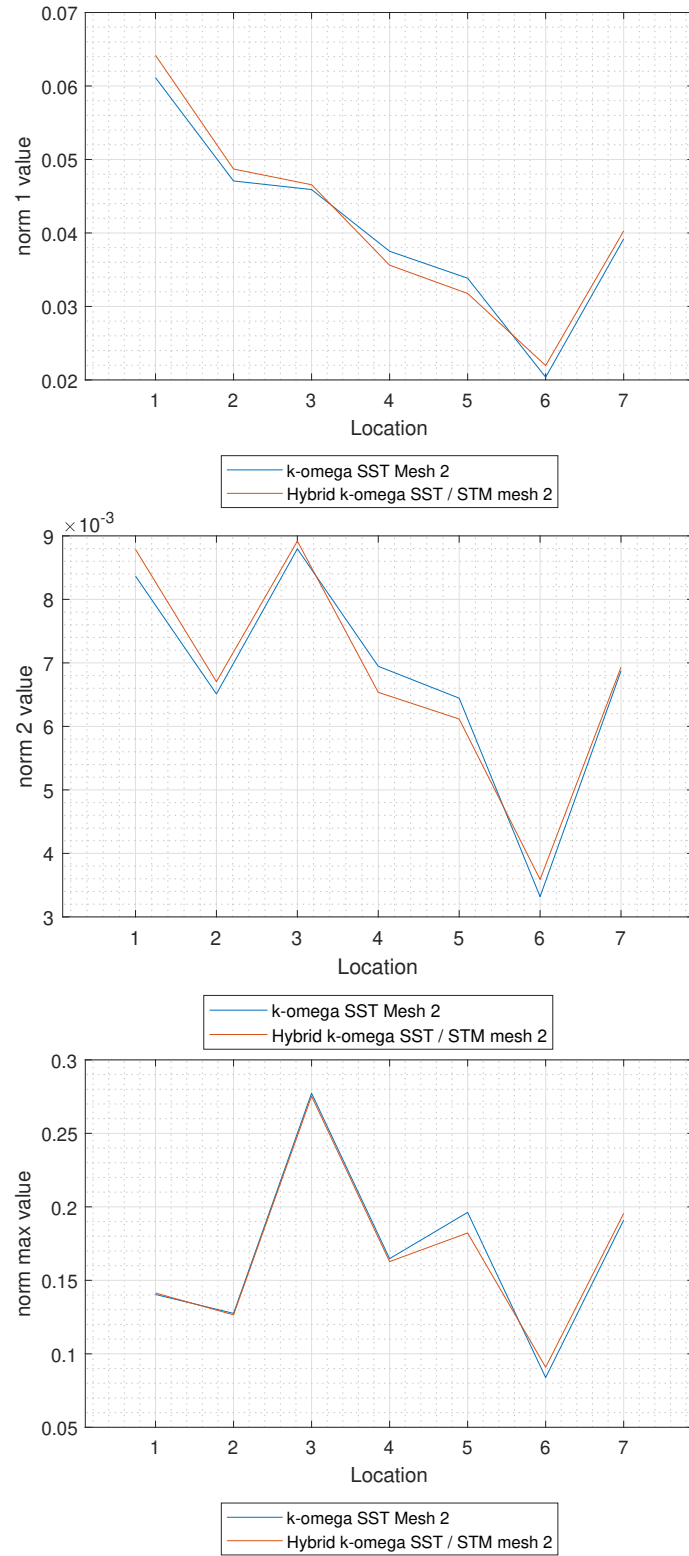


Figure 4.7:

- (a) Norm 1 comparison of errors from the turbulence models
- (b) Norm 2 comparison of errors from the turbulence models
- (c) Norm max comparison of errors from the turbulence models

4. Results and Discussion

The norm 2 errors in figure 4.7 (b) show the absolute errors with additional weighting towards larger absolute errors in the results. It can be immediately noted that the largest discrepancies in error can be observed at the same locations as in the norm 1 values in figure 4.7 (a), indicating that the main difference between the models comes from larger more isolated errors. In location 4 and 5, this seems to arise from the mid to trailing edge pressure coefficients and for location 1 is likely from the anisotropic models issues with modelling the larger lambda shock region. One noticeable feature of the norm 2 plots is that at location 3, there is a steep increase in the norm value, not present in the norm 1 plots. As noted previously, the third location sees significant struggle from both turbulence models to represent the initial front shock of the lambda shock system, which is similar for both models. Overall, again both models perform well with low norm 2 values indicating that the models are representing the experimental data well in most cases.

The norm max values plotted in figure 4.7 (c) help to assure the findings from the norm 2 and norm 1 plot distributions. An interesting initial observation is that the maximum error at the first location is actually the second lowest of all locations, whereas in figures 4.7 (a),(b), this location had among the highest norm values. This indicates that the high norm 1 and 2 values come from many smaller errors dominating the norms rather than large errors like location 3. Here, the maximum error seems to be approximately 0.275, which helps to show why its norm 2 value increased significantly when compared to norm 1. This leads to the conclusion that location 3 absolute errors largely originate from few larger errors. Both models maximum absolute errors are extremely close for almost all locations. It can be concluded that the majority or difference in absolute errors therefore are coming from smaller discrepancies between the models like the mid-trailing edge pressure modelling error difference between the anisotropic and K-Omega SST case. The anisotropic maximum error seems to be significantly less than the K-Omega SST model for location 5, this most likely comes from the enhanced turbulence representation by using an anisotropic approach.

4.2 Channel flow validation case

Channel flow cases are classic examples of validation cases for fluid dynamics, usually designed to test the wall modelling capabilities of a model. Here, the RANS data has been evaluated against a unique DNS case with the results gained from a code possible to run on any workstation. This comes with some assumptions made on the DNS side, but the data has been validated in the paper where the code was proposed (7) and so the data is reliable to use. Using DNS data over experimental data presents one major advantage in

4. Results and Discussion

that many flow variables like the stress tensor elements can be easily extracted from the results which may be more challenging to calculate or measure from an experiment. It is important to analyse the wall modelling for this case through the tensor elements but also to address the wider performance through velocity and other contours. In addition, in the book by Dr Könözy (2), it was stated that results were found to be best at very low y^+ values, and so some analysis of the mesh sensitivity can also be performed. Dr Könözy also proposes the use of different model constants for better near wall modelling and so the effects of these should also be quantified.

4.2.1 Reynolds stresses comparisons

Analysis of the Reynolds stresses are the key to validating this turbulence model due to its focus on the anisotropic tensor elements. A comparison of both anisotropic and isotropic tensor elements must be made to ensure the model is working effectively. The tensor elements can be compared to DNS data to analyse the differences. It is important to note that there may still be some significant difference between the two data sets due to the nature of comparing a steady RANS simulations to a DNS simulation. The difference may come from transient effects, from RANS averaging and assumptions or from the turbulence model itself, making it important to derive where these differences are arising from. Data has been compared for the three meshes and for adjusted wall modelling constants and split between isotropic and anisotropic elements. It is important to note as well that the DNS data takes averages of the Reynolds stress components over its 3m domain. With the same method being difficult to achieve in fluent and so the data was taken from a point where the friction Reynolds number was equal to that of the averaged DNS domain. This may result in some unavoidable differences between the two datasets.

Looking first at the isotropic components of the Reynolds stress tensor in figure 4.8 (a), each component has been plotted for the three meshes against the DNS data. Figure 4.8(a) shows the stream wise fluctuations in the velocity. It can be seen that for all three meshes, the peak value of the stress near the wall is under-represented. The coarse and medium mesh also seem to under-predict the fluctuations further from the wall and both meshes perform similarly overall which is likely due to them not refining the near-wall well as seen again in figure 4.13. The third mesh makes good improvement in the region further from the wall, only deviating from the DNS data by approximately 0.25. However, it still under-estimates the near wall region, indicating that an improvement in near wall modelling is needed, for example, with adjusted constants.

4. Results and Discussion

Looking at the vertical fluctuations perpendicular to the wall in figure 4.8 (b), all three meshes behave similarly in the near wall region but all over predict the near wall fluctuations. For the coarse and medium meshes, the fluctuations seem to dissipate at a much higher rate than the DNS data resulting in a significant under prediction of the value in the buffer layer. The finest mesh again improves on the capturing of the tensor elements, with a reduction in the sharpness in which the fluctuations fall off in the region further from the wall. Although, again, the dissipation is still too high and so the fluctuations finish below the DNS data. This may be fixed with a change of the model constants but is also a common drawback of eddy viscosity based turbulence models where they are often overly dissipative to aid conversion of results.

Finally, for the span wise fluctuations as seen in figure 4.8 (c), there is significant improvements in the models ability to capture the stress tensor element. While the coarse and middle meshes again have much higher dissipation of fluctuations further from the wall. All three meshes predict the initial peak near the wall well but still slightly over predict the peak value. The overall profiles of the vertical and horizontal Reynolds stress components are similar as in most turbulence models. The third mesh shows great improvement over its coarser counterparts with the dissipation in fluctuations still being higher than in the DNS data but overall providing comparable results.

4. Results and Discussion

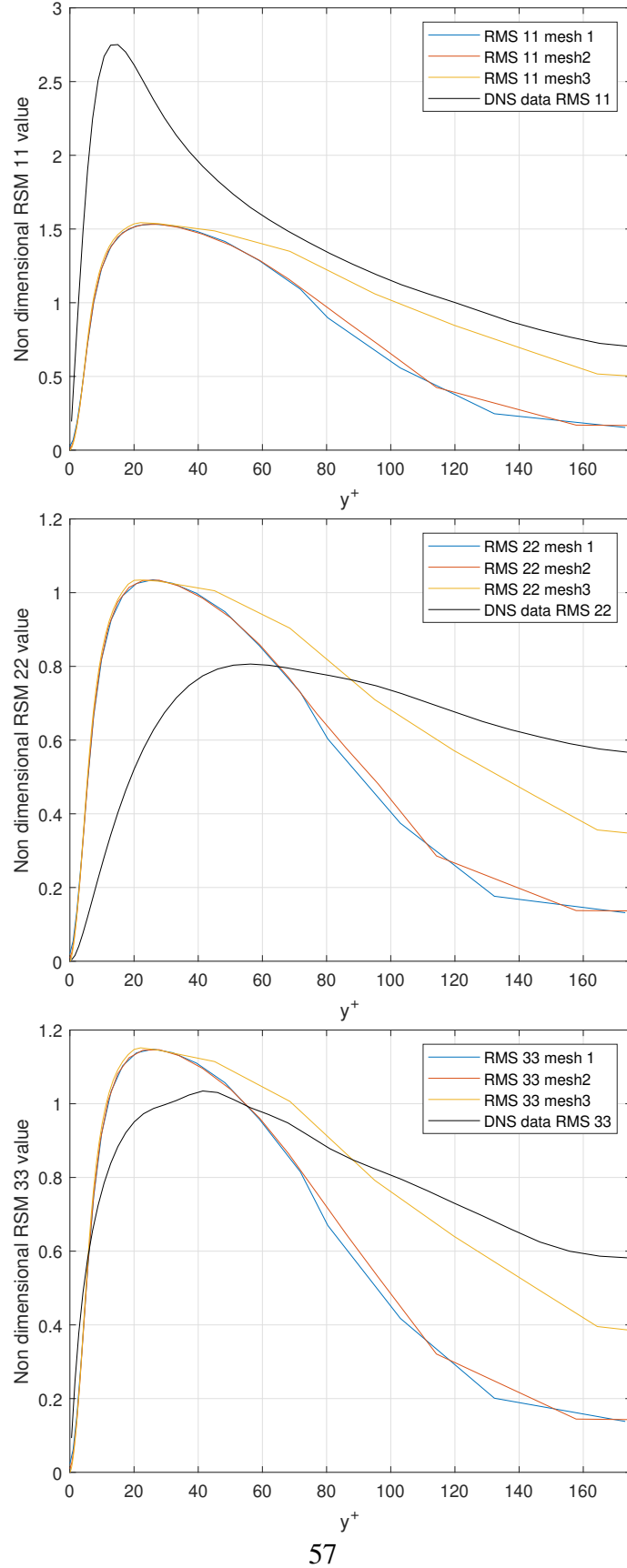


Figure 4.8:

- (a) Comparison of non dimensional RMS 11 Reynolds stress for channel flow
- (b) Comparison of non dimensional RMS 22 Reynolds stress for channel flow
- (c) Comparison of non dimensional RMS 33 Reynolds stress for channel flow

4. Results and Discussion

The anisotropic elements of the Reynolds stress tensor can be seen in figure 4.9. These are the elements usually assumed to be zero by most turbulence models. It is clear from looking at the DNS data that this is not the case in reality. First looking at the meshes handling of the $u'_1 u'_2$ or stream wise - vertical fluctuations. These fluctuations along with the stream wise-span wise fluctuations would be expected to be the strongest due to the driving flow being in one of the component directions. Figure 4.9 (a) shows the mesh and DNS data's representation of this stress tensor component. It can be seen that all three meshes capture the peak amplitude of the fluctuations well, but predict the peak to be slightly closer to the wall than in the DNS data. The fine mesh makes significant improvements over the coarser meshes, matching the dissipation of fluctuations extremely well compared to the DNS data.

The stream wise - span wise fluctuations prove difficult to model for this case, with a significant under-prediction across the whole boundary layer as seen in figure 4.9 (b). There seems to be little improvement across the three meshes indicating this is not a grid issue. Instead, this may come down to an issue with constants of the turbulence model where the contributions of this element is being under-represented.

The final anisotropic element comes from the vertical - span wise fluctuations seen in figure 4.9(C). As mentioned, this component would be expected to be smallest and this is the case here. Overall all three meshes present results significantly lower than the DNS data with the initial rise in fluctuations identifiable in the finest mesh but still below the DNS data. Again each mesh shows some improvement over the last but still lie well below the DNS data indicating that it may be an issue of model constants.

Overall, the model seems to be sensitive to the resolution of the mesh towards the wall. This is reiterated in Dr Könözy's work (2). This has its drawbacks as although this model seems to be able to predict anisotropic and isotropic elements well without introducing additional transport equations, there may still be some computational cost increase from having an extremely fine mesh at the wall. However, when compared to methods such as DNS, LES and other anisotropic RANS solutions such as RSM's, the computational cost from extra wall refinement is still much less than any other method.

4. Results and Discussion

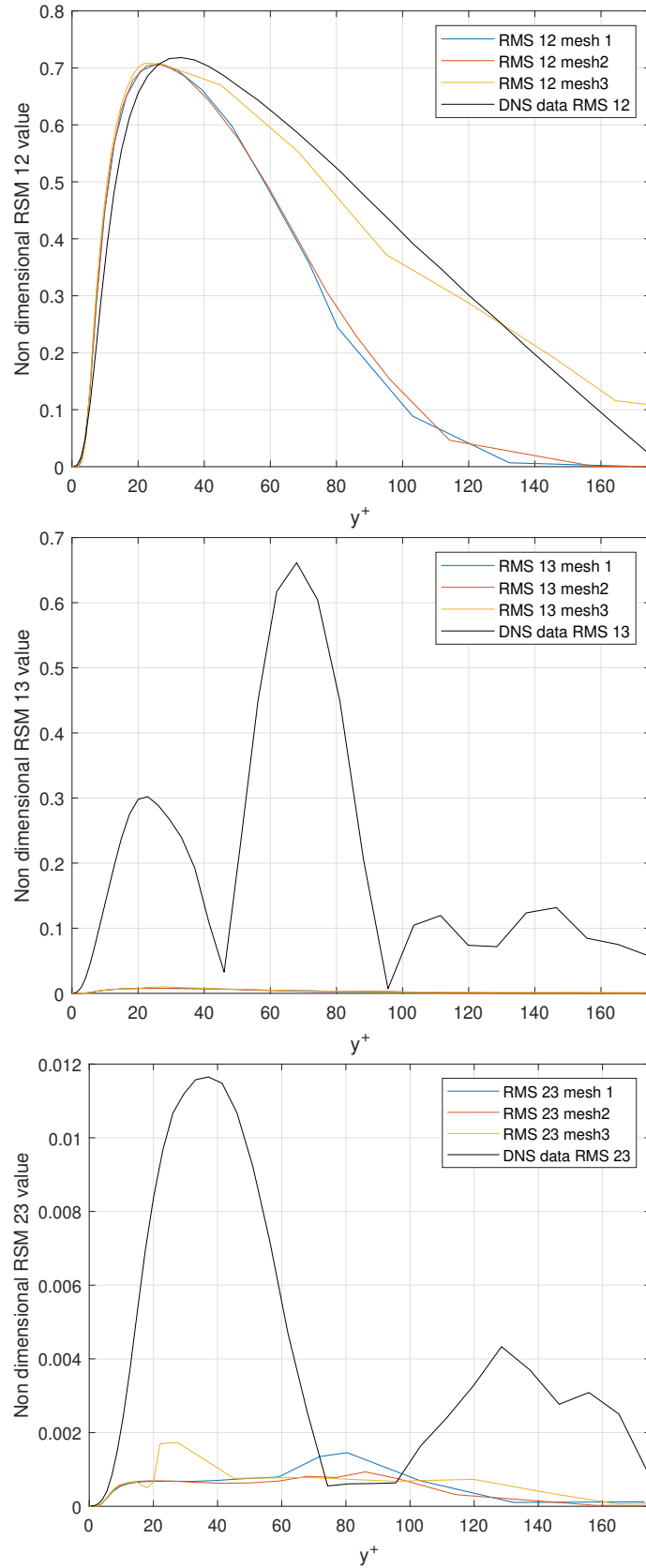


Figure 4.9:

- (a) Comparison of non dimensional RMS 12 Reynolds stress for channel flow
- (b) Comparison of non dimensional RMS 13 Reynolds stress for channel flow
- (c) Comparison of non dimensional RMS 23 Reynolds stress for channel flow

4. Results and Discussion

Dr Könözy suggests some adjusted constants to be used for high Reynolds number as well as for improvement of near wall modelling (2). For this, the near wall modelling constants have been used to help improve the behaviour of the model although it should be noted that the use of these constants may affect the values predicted further from the wall. Looking first at figure 4.10(a), the stream wise fluctuations are displayed for both sets of constants as well as the DNS data. It can be noted that the adjusted constants do improve the near wall predictions, with the peak values being closer to the DNS data. However, like the original constants, it does not capture the same profile of peak that the DNS fluctuations has. In addition, in the region further from the wall, the fluctuations are slightly over predicted but overall do still lie closer to the DNS values then the original constants case.

Figure 4.10(b) shows the vertical component of the fluctuations in velocity. Here, the adjusted constants show some improvement again over the original constants. The adjusted constants case models the peak fluctuations to an identical value compared to the DNS data although the peak location does seem to vary in its location, being slightly earlier than the DNS data. In addition to this, the new constants improve the dissipation somewhat with a reduction in the gradient in which the fluctuations drop off after their peak value. Although, due to the lower but more correct peak modelled, the final fluctuation value is still further off than the original constants. With this considered, it seems that the model constants may need to be blended based on wall height to further improve results.

Finally, figure 4.10 (c) shows the span wise variations in the velocity at the designated location. Here, again, the adjusted constants work to improve the data in line with the DNS data although it does still over predict the peak value and peaks to a maximum value too early. Both sets of constants seem to be overly dissipative for this case with the gradient exceeding that of the DNS data. This disparity seems to be common amongst all of the isotropic components, perhaps further tuning of constants of the model could be made to reduce the dissipation of the models especially with regards to the span wise and vertical Reynolds stress components.

4. Results and Discussion

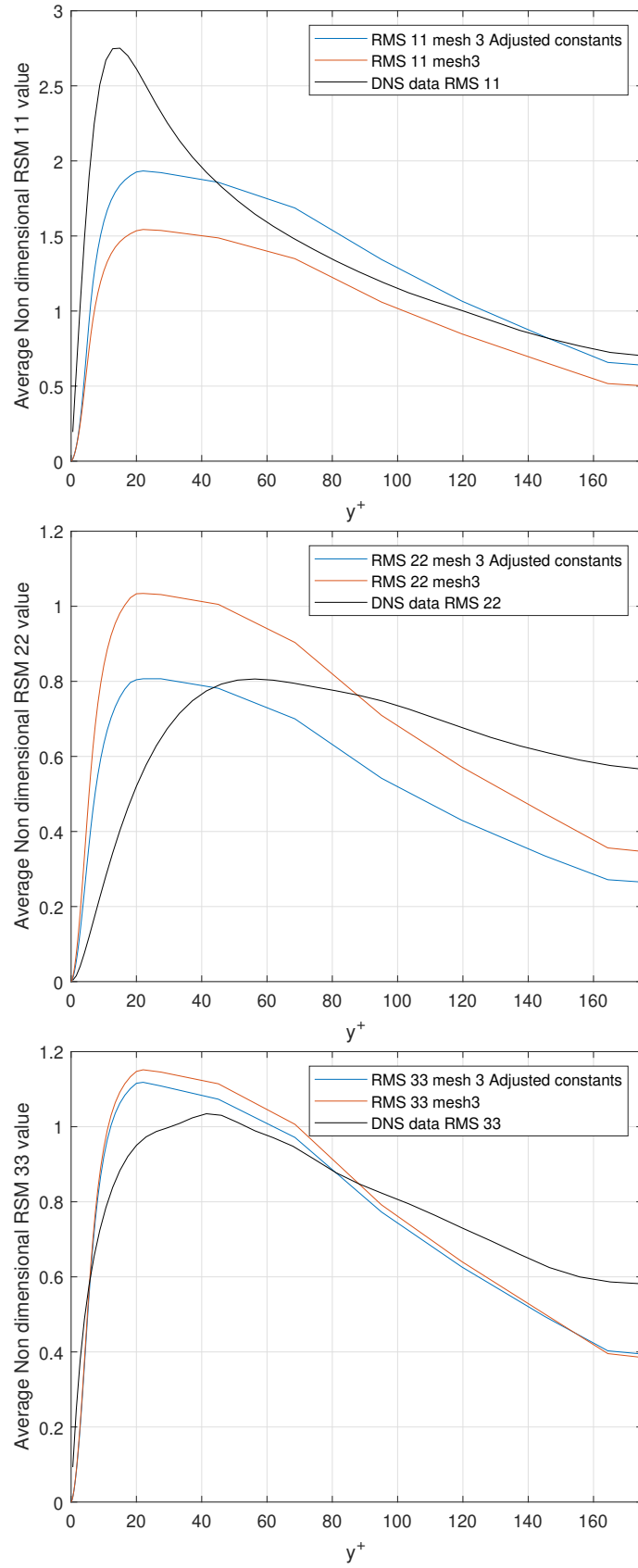


Figure 4.10:

- (a) Comparison of non dimensional RMS 11 Reynolds stress for channel flow
- (b) Comparison of non dimensional RMS 22 Reynolds stress for channel flow
- (c) Comparison of non dimensional RMS 33 Reynolds stress for channel flow

4. Results and Discussion

The anisotropic components of the Reynolds stress tensor can be seen plotted in figure 4.11. The Reynolds stress tensor elements can be seen for the original constants as well as for the adjusted constants. The stream wise - vertical fluctuations can be seen in figure 4.11(a) where both sets of constants show extremely good correlation with the DNS data. The adjusted constants seem to predict the initial peak value well but it is still predicted to be slightly too early but then predicts the initial dissipation in fluctuations extremely well. Both sets of constants seem to have a kink in which the dissipation rate of velocity fluctuations changes, differently to the relatively constant rate from the DNS data. This happens at the same location for both cases and so is most likely influenced by the mesh, perhaps more refinement of the region further from the wall may be required.

The stream wise - span wise fluctuations remain difficult for the model to represent even with the improved wall-modelling constants. This can be seen in figure 4.10(b) where there is some improvement in the values. It is small and still shows a significant reduction in fluctuations when compared to the DNS data. It remains unclear the exact reason for this lapse in the models ability to model this tensor element but it does not appear to be related to the adjusted constants. It could still be a factor of the mesh refinement as this mesh sees a maximum y^+ of 0.14 whereas the ideal would be for a value of 0.02 as mentioned by Könözy (2). Although from figure 4.12, the majority of the mesh does seem to have a y^+ of around 0.024 and so it should still be very well refined at the wall.

The span wise - vertical fluctuations seen in figure 4.11(c) show some distinct differences between the two model constant approaches. The adjusted constants seem to significantly increase the peak fluctuation value over the original constants although it peaks to a higher value than that of the DNS predication. In addition, as expected in the region further from the wall, the adjusted constants fall short of predicting the small fluctuations shown in the DNS data. This may indicate a finer mesh further from the wall may be needed.

Overall, the adjusted constants do seem to improve the modelling of the Reynolds stress tensor elements for the most part with excellent correlation for the isotropic elements. There seems to be some need for improvement for the region further from the wall, perhaps a blending value for the constants similar to that which the K-Omega SST model uses for blending the influence of the K-Omega SST model to K-Epsilon model based on wall distance.

4. Results and Discussion

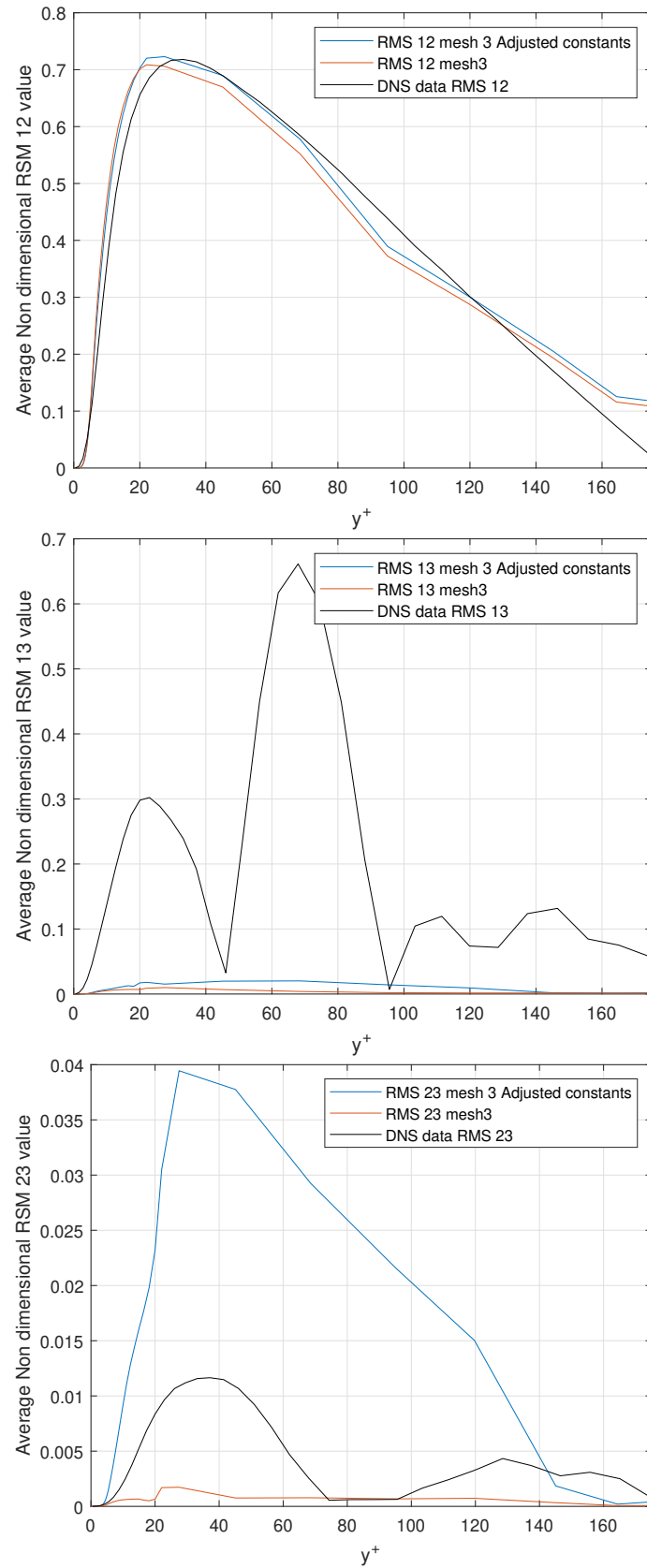


Figure 4.11:

- (a) Comparison of non dimensional RMS 12 Reynolds stress for channel flow
- (b) Comparison of non dimensional RMS 13 Reynolds stress for channel flow
- (c) Comparison of non dimensional RMS 23 Reynolds stress for channel flow

4. Results and Discussion

4.2.2 Supporting contours comparisons

4.2.2.1 Turbulence capturing

The y plus value given by the mesh at the wall is a crucial indicator for the meshes ability to handle wall modelling. A y plus ≤ 1 should provide the resolution for the turbulence model to solve the near wall flow without the need for wall models. This is crucial for this case as it is the near wall turbulence where the highest anisotropy appears in the flow. Studies by Könözy (2) show that for best results with the hybrid anisotropic model, a y plus of 0.2 is required. Figure 4.12 shows the y plus distribution on each wall. It can be seen that in all of the simulations the peak y plus value appears at the inlet, indicating a slight sensitivity at the inlet for the mesh. Figure 4.12 (a) is the coarsest mesh where it can be seen that the maximum y plus at the inlet is at 2.35, which is too high to represent the flow without wall models initially and so a further refinement is needed. The wall resolution seems to be strong still for even the coarse mesh with it reaching a minimum y plus of 0.49. Looking at the middle mesh in figure 4.12 (b), the mesh is much finer with a minimum y plus of 0.28, however, the maximum y plus is still too high at 1.7 indicating further refinement is crucial and shows why the final mesh was chosen. The fine mesh in figure 4.12 (c) shows a strong y plus value for the whole domain. With a maximum value of 0.14, being lower than the minimum of the previous mesh and a minimum value of 0.024, the mesh is very fine at the wall however, still does not reach the optimum value proposed by Könözy. This however, was the finest the mesh could be made due to issues with running the simulations on a more powerful HPC. Therefore, the fine mesh was chosen for running further simulations.

4. Results and Discussion

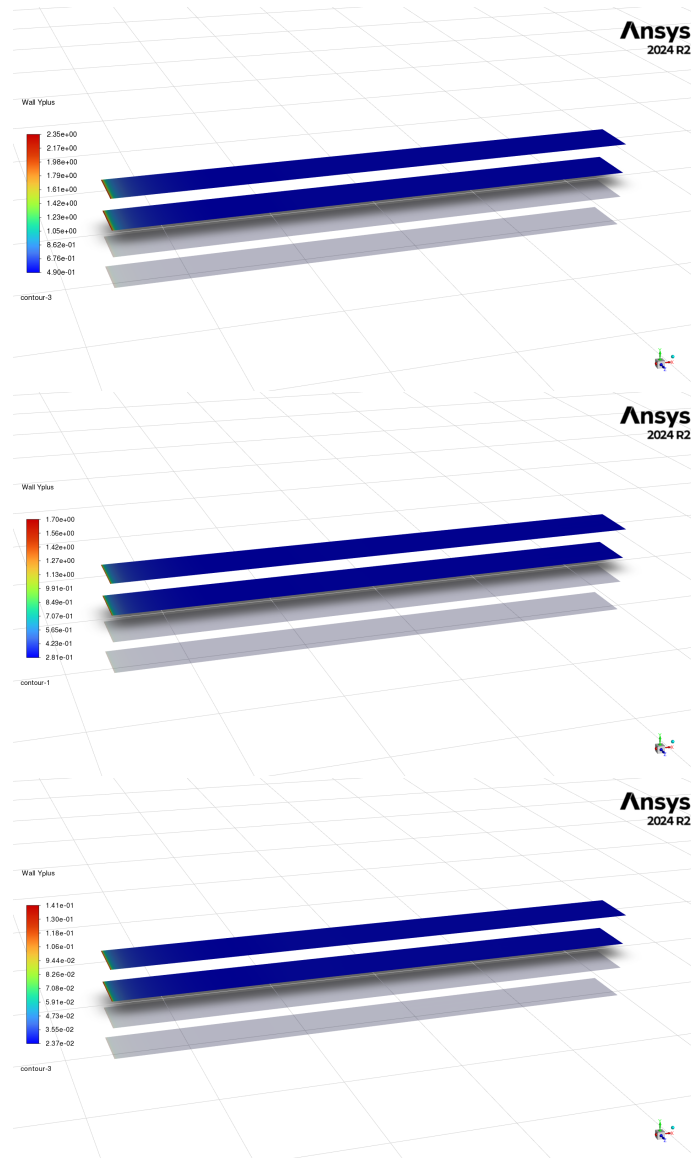


Figure 4.12:

- (a) y plus contour for mesh 1
- (b) y plus contour for mesh 2
- (c) y plus contour for mesh 3

As mentioned for the Onera M6 case, normalised Q criterion can be a useful tool for looking at the vortical and turbulent flows in the domain and how well the chosen mesh and turbulence models are capturing such structures. For this case normalised Q-criterion is plotted in the channel with turbulent viscosity contours plotted onto this iso-surface. The contours for the coarse, medium and fine mesh as well as for the adjusted constants

4. Results and Discussion

case can all be seen in figure 4.13.

Looking first at the the coarsest mesh in figure 4.13 (a), it is immediately noticeable that the level of resolution of the turbulent structures is low with only the larger scales fully resolved here with it being noticeably smeared out for the most part. In addition, the boundary layer seems to transition from laminar to turbulent much later in the channel for this flow case. In addition, when compared with the other results, the turbulent viscosity seems to be much smoother, indicating that this is an artificial viscosity from some degree of wall modelling rather than fine near wall modelling. This is all largely expected as this is the finest of the three meshes.

Next, comparing the second mesh to the first in figure 4.13 (a) (b), a noticeable difference can be observed in the normalised Q-criterion iso-surface. In the middle mesh, the boundary layer seems to transition to turbulent much sooner than in the coarse mesh, identifiable by the small structures in the iso-surface. This coincides with a much clearer turbulent viscosity layering in the turbulent boundary layer in addition to clearer turbulent structures such as the vortex cores in the boundary layer. All of this indicates that the model is capturing the near wall fluctuations much better with the finer mesh.

The finer mesh shows a sharp improvement over both of the previous cases with regards to the boundary layer modelling. Comparing the fine mesh to the middle mesh in figure 4.13(b) (c), it is immediately clear that the onset of turbulence happens much closer to the inlet for this case. In addition, the vortical structures are captured here at an excellent resolution for a RANS case with small and large eddies visible showing the cascading energy expected in turbulent boundary layer flow. In addition, the vortex roll up and instabilities can be observed in the near wall region. Each of these indicates that the near-wall region is well refined here and is capturing the small fluctuations in velocity to a great resolution. The turbulent viscosity contours here show, as expected, with initially low values where the flow is laminar or transitioning before becoming much higher towards the outlet where the turbulence is decaying into smaller length scales, inducing more mixing effects and so a higher turbulent viscosity.

Finally, it is useful to see if the adjusted constants have helped for the near wall modelling of the case, which can be compared in figure 4.13 (c) (d). There is a noticeable difference between the two models, indicating that the model is sensitive to these values. Turbulence onset seems to be sooner for the adjusted constants case but by a much smaller difference than between the medium and fine mesh. As well as this, the turbulent viscosity in the boundary layer seems to increase sooner, indicating a faster decay into smaller turbulent

4. Results and Discussion

structures, which can be seen as well when looking at the iso-surface to see the size of the vortex cores. The increase in turbulent viscosity and varying turbulent structures likely comes from the constants. These constants represent the weighting of the anisotropic components of the stress tensor and so an increase in their values will inevitably change the scales and structure of the turbulence. It is useful to view the physical difference in turbulent structures however, it is unclear which is the most accurate representation of the turbulence without comparing to the experimental results and so judgement will be made when comparing Reynolds tensor components in figure 4.11.

A common feature of all the simulations seems to be a region picked up by the normalised Q-criterion iso-surface at the outlet of increased strain in the flow. This is a likely consequence of the outlet flow being set to prevent reverse flow.

4. Results and Discussion

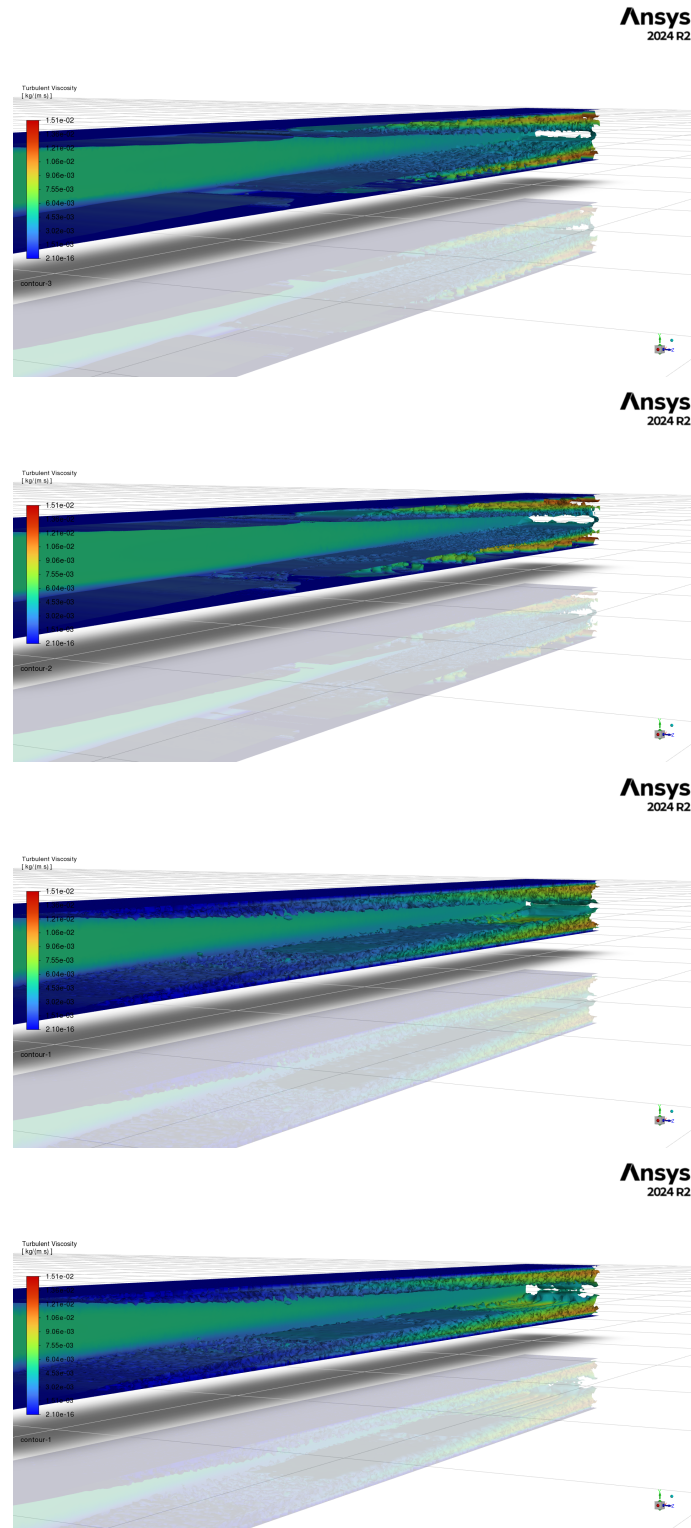


Figure 4.13:

- (a) Turbulent viscosity contour on Q-criterion iso-surface for mesh 1
- (b) Turbulent viscosity contour on Q-criterion iso-surface for mesh 2
- (c) Turbulent viscosity contour on Q-criterion iso-surface for mesh 3
- (d) Turbulent viscosity contour on Q-criterion iso-surface for mesh 3 with adjusted constants

4. Results and Discussion

4.2.2.2 Velocity contours

Velocity contours for this case can be used to view the boundary layer growth along the walls and help to view how each mesh is capturing the boundary layer growth. Looking at figure 4.14, the velocity contours for each mesh, including the case with adjusted constants for wall modelling, can be viewed. Overall, all four cases seem to model the boundary layer size extremely similarly, one noticeable difference being with regards to the boundary layer height being slightly higher towards the outlet for all but the coarsest mesh. In addition, looking extremely closely at the edge of the boundary layer for all three cases, the finer meshes seem to have more fluctuations in the turbulent layer. This is especially prevalent in the adjusted constants sub figure. This shows the effects of further refinement in its ability to capture smaller scales of turbulence.

4. Results and Discussion

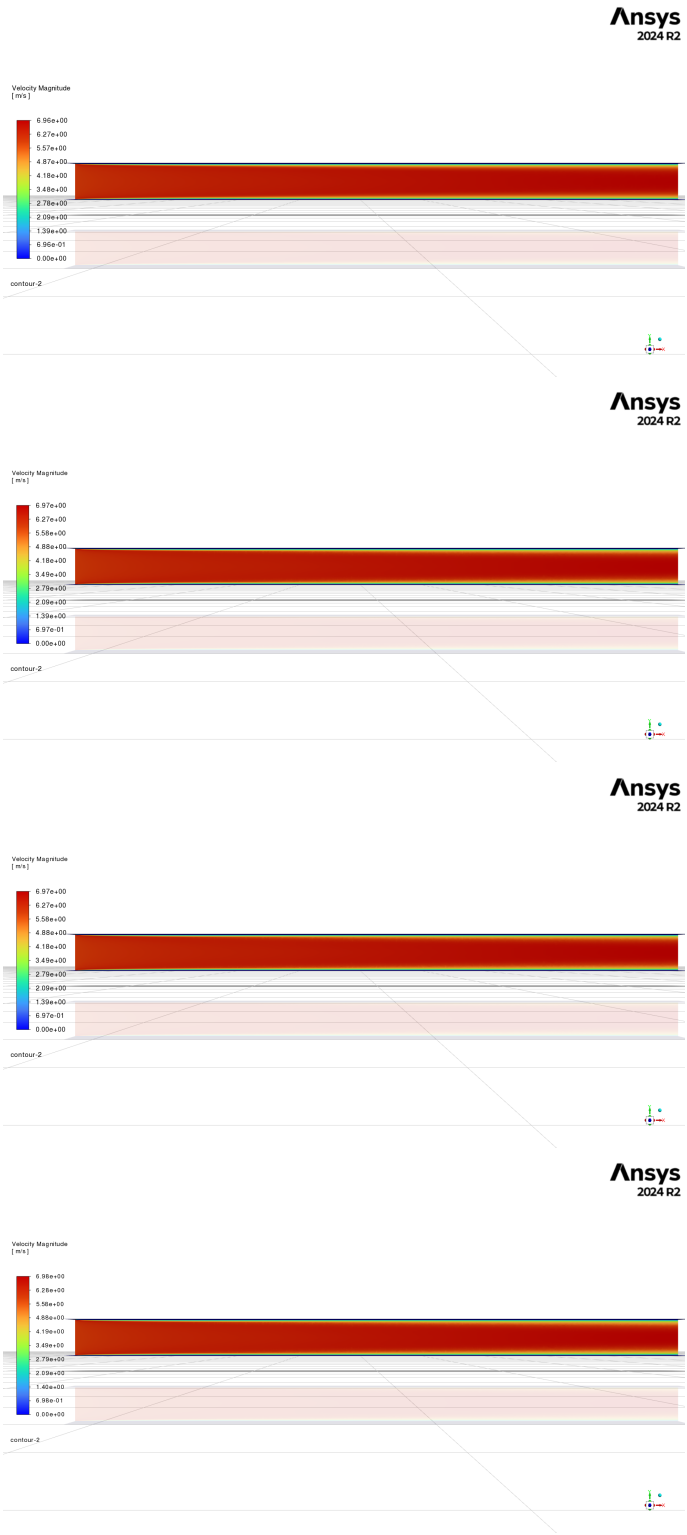


Figure 4.14:
(a) y plus contour for mesh 1
(b) y plus contour for mesh 2
(c) y plus contour for mesh 3
(d) y plus contour for mesh 3 with adjusted constants

4. Results and Discussion

4.3 Conclusion and Future Work

4.3.1 Conclusion

To conclude the study of the Onera M6 case, it is clear that while both models perform extremely well for this test case, the anisotropic hybrid K-Omega SST/STM model (1) (2) helps to capture some flow features which the isotropic nature of the K-Omega SST model cannot represent. This is mostly in regards to wing tip vortex interactions with the shock formation. In addition, the shear layer or turbulent wake produced by the small detachment of flow post-shock is better represented by the anisotropic model. The K-Omega SST model seems to outperform the anisotropic model in some regards, for example, the dissipation terms of the anisotropic model seem to not provide enough damping when looking at the pressure coefficients. Therefore, it may be concluded that more work into model tuning may be needed for transonic / supersonic flow regimes.

With regards to the channel flow case, in general, the model does a good job at modelling the Reynolds stress tensor elements. With the use of adjusted constants, the model represents the isotropic and stream wise - vertical components in excellent correlation to the experimental data. The model shows reasonable representation of the remaining two Reynolds stresses. This can be linked to a variety of factors including the mesh discretisation. Previous best results were found at a y^+ of 0.2 but due to limitations in mesh size, the final mesh fell slightly short of this value. In addition, some further tuning of the model constants seems to be required to help with matching the data to DNS values. With regards to the DNS data, the DNSlab case provides a useful tool for modelling turbulent flow in all of its length scales and provides a useful validation case.

Overall, the hybrid anisotropic K-Omega SST/STM turbulence model shows marked improvement over the K-Omega SST model by itself in modelling physically accurate turbulent flow. Although the model works well for the most part, some further work is needed for improving the model's capabilities. By representing the full Reynolds stress tensor, including the anisotropic elements with little impact on the computational cost, the model has applications to any flow cases where there is a presence of walls, vortices and turbulent wakes, therefore, making it a vital improvement in our ability to model flows in a physically correct manner.

4.3.2 Future Work

While the model works well at representing the flow in a more physical nature, there is room for improvement in some areas. The first may be to look into a blending function

4. Results and Discussion

for the model constants to further improve the Reynolds stress tensor elements prediction in the near and far-wall regions. In addition, some investigation should take place into the models dissipative nature as it seems to predict a faster decay in fluctuations than the DNS data for most components of the Reynolds stress tensor. In addition, some further work could be performed to validate the model over a wider range of validation cases for example a fully supersonic test case. Further validation cases may provide insight into whether the model constants need further tuning to better represent the Reynolds stress tensor elements for a variety of different flow conditions. Another possible test case could be to test the model in a modified form as a sub grid scale model for LES as these models are typically isotropic similarly to RANS. Overall, this model shows promise at allowing engineers and researchers to model turbulent flows in a more physical and accurate manor and so further investigation and optimisation of the model would help in this process.

Bibliography

- [1] Könözsy L. A new hypothesis on the anisotropic Reynolds stress tensor for turbulent flows. vol. 1. Springer; 2019.
- [2] Könözsy L. A New Hypothesis on the Anisotropic Reynolds Stress Tensor for Turbulent Flows: Volume II: Practical Implementation and Applications of an Anisotropic Hybrid K-omega Shear-Stress Transport/Stochastic Turbulence Model. vol. 125. Springer Nature; 2020.
- [3] Chen CJ, Jaw SY. Fundamentals of turbulence modelling; 1997.
- [4] Nelkin M. In What Sense Is Turbulence an Unsolved Problem? Science. 1992;255(5044):566-70. Available at: <https://www.science.org/doi/abs/10.1126/science.255.5044.566>.
- [5] Navier CL. Mémoire sur les lois du mouvement des fluides. Mémoires de l'Académie Royale des Sciences de l'Institut de France. 1822;6:389-440. Available at: <https://gallica.bnf.fr/ark:/12148/bpt6k90194f/f389.item>. Presented March 18, 1822. Published 1827.
- [6] Stokes GG. On the theories of the internal friction of fluids in motion and of the equilibrium and motion of elastic solids. Transactions of the Cambridge Philosophical Society. 1845;8:287-319. Available at: <https://www.cambridge.org/core/books/abs/mathematical-and-physical-papers/on-the-theories-of-the-internal-friction-of-fluids-in-motion-and-of-the-equilibrium-and-motion-of-elastic-solids/03959CB979233BCA95EB128CC5FF6D41>.
- [7] Verstappen R, Veldman A. Direct Numerical Simulation of Turbulence at Lower Costs. Journal of Engineering Mathematics. 1997 10;32.
- [8] Domingo P, Vervisch L. Recent developments in DNS of turbulent combustion. Proceedings of the Combustion Institute. 2023;39(2):2055-76. Available at: <https://www.sciencedirect.com/science/article/pii/S1540748922000864>.
- [9] Malinverno G, Alberio JB, SanMartin JL. Quantum computing CFD simulations: state of the art. Materials Research Proceedings. 2024;37.

Bibliography

- [10] Syamlal M, Copen C, Takahashi M, Hall B. Computational Fluid Dynamics on Quantum Computers;. Available at: <https://arc.aiaa.org/doi/abs/10.2514/6.2024-3534>.
- [11] Deardorff JW. A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers. *Journal of Fluid Mechanics*. 1970;41(2):453–480.
- [12] Moeng CH. A large-eddy-simulation model for the study of planetary boundary-layer turbulence. *Journal of Atmospheric Sciences*. 1984;41(13):2052-62.
- [13] Smagorinsky J. General circulation experiments with the primitive equations: I. The basic experiment. *Monthly weather review*. 1963;91(3):99-164.
- [14] Bou-Zeid E. Challenging the large eddy simulation technique with advanced a posteriori tests. *Journal of Fluid Mechanics*. 2015;764:1–4.
- [15] Luo XY, Hinton JS, Liew TT, Tan KK. LES modelling of flow in a simple airway model. *Medical Engineering Physics*. 2004;26(5):403-13. Available at: <https://www.sciencedirect.com/science/article/pii/S1350453304000347>.
- [16] Bensow RE, and CF. On the justification and extension of mixed models in LES. *Journal of Turbulence*. 2007;8:N54. Available at: <https://doi.org/10.1080/14685240701742335>.
- [17] Smits AJ, McKeon BJ, Marusic I. High-Reynolds Number Wall Turbulence [Journal Article]. *Annual Review of Fluid Mechanics*. 2011;43(Volume 43, 2011):353-75. Available at: <https://www.annualreviews.org/content/journals/10.1146/annurev-fluid-122109-160753>.
- [18] Balaras E, Benocci C, Piomelli U. Two-layer approximate boundary conditions for large-eddy simulations. *AIAA Journal*. 1996;34(6):1111-9.
- [19] Cabot WH, Moin P. Approximate wall boundary conditions in the large-eddy simulation of high Reynolds number flow. *Flow, Turbulence and Combustion*. 2000;63(1):269-91.
- [20] Yang XIA, Griffin KP. Grid-point and time-step requirements for direct numerical simulation and large-eddy simulation. *Physics of Fluids*. 2021;33(1):015108.
- [21] Fu L, Griffin KP. A new ODE-based turbulence wall model accounting for pressure gradient and Reynolds number effects. *arXiv preprint arXiv:201004097*. 2020.
- [22] Spalart PR, Jou WH, Strelets M, Allmaras SR. Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach. Liu C, Liu Z, (eds.). *Advances in DNS/LES*. Columbus, OH: Greyden Press; 1997. p. 137-47.
- [23] SPALART P, ALLMARAS S. A one-equation turbulence model for aerodynamic flows;. Available at: <https://arc.aiaa.org/doi/abs/10.2514/6.1992-439>.

Bibliography

- [24] Spalart PR. Detached-Eddy Simulation. *Annual Review of Fluid Mechanics*. 2009;41:181-202.
- [25] Heinz S, Fagbade A. A Critical Review of Hybrid RANS-LES Concepts: Continuous Eddy Simulation Versus Classical Methods;. Available at: <https://arc.aiaa.org/doi/abs/10.2514/6.2025-2214>.
- [26] Spalart PR, Deck S, Shur ML, Squires KD, Strelets MK, Travin AK. A new version of detached-eddy simulation, resistant to ambiguous grid densities. *Theoretical and Computational Fluid Dynamics*. 2006;20(3):181-95.
- [27] Shur ML, Spalart PR, Strelets MK, Travin AK. A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities. *International Journal of Heat and Fluid Flow*. 2008;29(6):1638-49.
- [28] Reynolds O. IV. On the dynamical theory of incompressible viscous fluids and the determination of the criterion. *Philosophical Transactions of the Royal Society of London (A)*. 1895;186:123-64. Available at: <https://royalsocietypublishing.org/doi/abs/10.1098/rsta.1895.0004>.
- [29] Wilcox D. *Turbulence Modeling for CFD (Third Edition) (Hardcover)*; 2006.
- [30] Spalart PR. Strategies for turbulence modelling and simulations. Rodi W, Laurence D, (eds.). *Engineering Turbulence Modelling and Experiments 4*. Oxford: Elsevier Science Ltd; 1999. p. 3-17. Available at: <https://www.sciencedirect.com/science/article/pii/B9780080433288500011>.
- [31] Obermeier F. Prandtl's Mixing Length Model - Revisited. *PAMM*. 2006;6(1):577-8. Available at: <https://onlinelibrary.wiley.com/doi/abs/10.1002/pamm.200610269>.
- [32] VAN DRIEST ER. On Turbulent Flow Near a Wall. *Journal of the Aeronautical Sciences*. 1956;23(11):1007-11. Available at: <https://doi.org/10.2514/8.3713>.
- [33] Bazargan M, Mohseni M. Algebraic zero-equation versus complex two-equation turbulence modeling in supercritical fluid flows. *Computers Fluids*. 2012;60:49-57. Available at: <https://www.sciencedirect.com/science/article/pii/S0045793012000771>.
- [34] Boussinesq JV. *Essai sur la théorie des eaux courantes*. Paris: Imprimerie Nationale; 1877. Available at: <https://gallica.bnf.fr/ark:/12148/bpt6k4336793>.
- [35] Prandtl L. Report on the studies of developed turbulence. Third International Congress for Applied Mechanics. Stockholm; 1925. Original work in German, later cited in turbulence modeling literature; see also Schlichting and Gersten, *Boundary-Layer Theory*.

Bibliography

- [36] Launder BE, Spalding DB. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*. 1974;3(2):269-89. Available at: <https://www.sciencedirect.com/science/article/pii/0045782574900292>.
- [37] Launder B, Morse A, Rodi W, Spalding D. Prediction of free shear flows: a comparison of the performance of six turbulence models. *NASA Langley Res Center Free Turbulent Shear Flows*, Vol 1. 1973.
- [38] Improvement of Standard K-Epsilon Turbulence Model for Round Free Jets by Adjusting Closure Coefficients. vol. Volume 9: *Fluids Engineering of ASME International Mechanical Engineering Congress and Exposition*; 2023. Available at: <https://doi.org/10.1115/IMECE2023-115131>.
- [39] Chen YS, Kim SW. Computation of turbulent flows using an extended k-epsilon turbulence closure model. *NASA*; 1987.
- [40] Wilcox DC. Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal*. 1988;26(11):1299-310. Available at: <https://doi.org/10.2514/3.10041>.
- [41] F R Menter MK, Langtry R. Ten Years of Industrial Experience with the SSTurbulence Model. *ANSYS*;
- [42] Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*. 1994;32:1598-605. Available at: <https://api.semanticscholar.org/CorpusID:120712103>.
- [43] Menter FR. Improved two-equation k-omega turbulence models for aerodynamic flows; 1992.
- [44] Younoussi S, Ettaouil A. Calibration method of the k- ω SST turbulence model for wind turbine performance prediction near stall condition. *Heliyon*. 2024;10(1).
- [45] Gutiérrez Amo R, Zamponi R, Ragni D, Llorente E, Aranguren P. On the extension of kSST corrections to predict flow separation on thick airfoils with leading-edge roughness. *Wind Energy*. 2023 05;26:n/a-n/a.
- [46] Mohotti D, Wijesooriya K, da Costa DD. Comparison of Reynolds Averaging Navier-Stokes (RANS) turbulent models in predicting wind pressure on tall buildings. *Journal of Building Engineering*. 2019;21:1-17. Available at: <https://www.sciencedirect.com/science/article/pii/S2352710218302997>.
- [47] ANSYS. Modelling turbulent flow, introductory FLUENT training; 2008. Available at: <https://www.slideserve.com/gregorylee/modeling-turbulent-flows-powerpoint-ppt-presentation>.

Bibliography

- [48] Uncertainty Quantification of Spalart–Allmaras Turbulence Model Coefficients for Simplified Compressor Flow Features; 2020.
- [49] Spalart PR, Garbaruk AV. A new “ λ_2 ” term for the spalart–allmaras turbulence model, active in axisymmetric flows. *Flow, Turbulence and Combustion*. 2021;107(2):245-56.
- [50] Paciorri R, Sabetta F. Compressibility Correction for the Spalart–Allmaras Model in Free-Shear Flows. *Journal of Spacecraft and Rockets*. 2003;40(3):326-31. Available at: <https://doi.org/10.2514/2.3967>.
- [51] Speziale CG, Sarkar S, Gatski TB. Modelling the pressure-strain correlation of turbulence: an invariant dynamical systems approach. *Journal of Fluid Mechanics*. 1991;227:245-72.
- [52] Launder BE, Reece GJ, Rodi W. Progress in the development of a Reynolds-stress turbulence closure. *Journal of Fluid Mechanics*. 1975;68(3):537-66.
- [53] Lee J, Sung HJ, Kim SK. Comparison of two-equation model and Reynolds stress models with experimental data for the three-dimensional turbulent boundary layer in a 30° bend. *International Journal of Heat and Fluid Flow*. 2000;21(4):374-83.
- [54] Spalart PR. Philosophies and fallacies in turbulence modeling. *Progress in Aerospace Sciences*. 2015;74:1-15. Available at: <https://www.sciencedirect.com/science/article/pii/S037604211400102X>.
- [55] Vitillo F, Galati C, Cachon L, Laroche E, Millan P. An anisotropic shear stress transport (ASST) model formulation. *Computers & Mathematics with Applications*. 2015 Nov;70(9):2238-51.
- [56] Wallin S, Johansson AV. An explicit algebraic Reynolds stress model for incompressible and compressible turbulent flows. *Journal of Fluid Mechanics*. 2000 Jan;403:89-132.
- [57] Dreeben TD, Pope SB. Probability density function and Reynolds-stress modeling of near-wall turbulent flows. *Physics of Fluids*. 1997 Jan;9(1):154-63.
- [58] Könözsy L. Three-Dimensional Simulations with an Anisotropic Hybrid k- ω SST/STMA Approach. *Cham : Springer International Publishing*; 2021. p.359 – 404.
- [59] Laufer J. The structure of turbulence in fully developed pipe flow. NASA; 1954.
- [60] Teschner TR. A comparative study between winglet and raked wingtip wing configurations; 2013.
- [61] Lawson NJ, Jacques H, Gautrey JE, Cooke AK, Holt JC, Garry KP. Jetstream 31 national flying laboratory: Lift and drag measurement and modelling. *Aerospace Science and Technology*. 2017;60:84-95.

Bibliography

- [62] Mokrane W. A Turbulent Hybrid Model to Simulate a Partially Pressurized Flow. Çiner A, Khan MF, Kallel A, Rodrigo-Comino J, Parise M, Barzegar R, et al., (eds.). Recent Research on Environmental Earth Sciences, Geomorphology, Soil Science, Paleoclimate, and Karst. Cham: Springer Nature Switzerland; 2023. p. 115-7.
- [63] Könözy L. Two-Dimensional Simulations with an Anisotropic Hybrid k- ω SST/STMA Approach. Cham : Springer International Publishing; 2021. p. 215 – 357. Available at : https://doi.org/10.1007/978-3-030-60603-9_4.
- [64] Schmitt V, Charpin F. 138. Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers. AGARD; 1979. Experimental Data Base for Computer Program Assessment. Report of the Fluid Dynamics Panel Working Group 04.
- [65] Shah MZBM, Basuno B, Abdullah AB. Comparative Study on Several Type of Turbulence Model Available in ANSYS-Fluent Software for ONERA M6 Wing Aerodynamic Analysis. Journal of Advanced Mechanical Engineering Applications. 2020;1(1):0-0. Available at: <https://doi.org/10.30880/jamea.2020.01.01.002>.
- [66] NASA Langley Research Center, Turbulence Modeling Resource. Grids – 3D ONERA M6 Wing for Turbulence Model Numerical Analysis; 2021. https://turbmodels.larc.nasa.gov/onerawingnumerics_grids.html. Available at : https://turbmodels.larc.nasa.gov/onerawingnumerics_grids.html. Last updated : November 10, 2021.
- [67] Ekaterinaris JA. High-order accurate, low numerical diffusion methods for aerodynamics;. Available at: <https://www.sciencedirect.com/science/article/pii/S0376042105000473>.
- [68] Li Wang J, TESK W, Kyle Anderson. High-order Methods for Solutions of Three-dimensional Turbulent Flows;. Available at: <https://arc.aiaa.org/doi/pdf/10.2514/6.2013-856>.
- [69] Mavriplis DJ. Revisiting the Least-Squares Procedure for Gradient Reconstruction on Unstructured Meshes. National Institute of Aerospace; 2003. Available at: https://scientific-sims.com/cfdlab/Dimitri_Mavriplis/HOME/assets/papers/aiaa20033986.pdf.
- [70] Vuorinen V, Keskinen K. DNSLab: A gateway to turbulent flow simulation in Matlab. Computer Physics Communications. 2016;203:278-89. Available at: <https://www.sciencedirect.com/science/article/pii/S0010465516300388>.
- [71] Vuorinen V, Keskinen K. DNSLab: A gateway to turbulent flow simulation in Matlab. Computer Physics Communications. 2016;203:278-89. Available at: <https://www.sciencedirect.com/science/article/pii/S0010465516300388>.
- [72] Vuorinen V, Keskinen JP, Duwig C, Boersma BJ. On the implementation of low-dissipative Runge–Kutta projection methods for time dependent flows using OpenFOAM®. Computers & Fluids. 2014;93:153-63.

Bibliography

- [73] Raje P, Sinha K. Formulation of advanced SST turbulence model for shock-boundary layer interaction; 2021. .

4.4 Appendices

4.4.1 Appendix A: