Numerical study of cargo bay turbulence of a transport aircraft

Pang, Samuel Kheng Hong

2012

Pang, S. K. H. (2012). Numerical study of cargo bay turbulence of a transport aircraft. Master's thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/50538>

<https://doi.org/10.32657/10356/50538>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

**NUMERICAL STUDY OF CARGO BAY TURBULENCE
OF A TRANSPORT AIRCRAFT**

PANG KHENG HONG SAMUEL

**SCHOOL OF MECHANICAL AND AEROSPACE
ENGINEERING**

2012

NUMERICAL STUDY OF CARGO BAY TURBULENCE OF A TRANSPORT AIRCRAFT

PANG KHENG HONG SAMUEL

School of Mechanical and Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in fulfillment of the requirement for the degree of
Master of Engineering

2012

Abstract

The Lockheed C-130 Hercules is arguably one of the most versatile fixed-wing transport aircraft available in any modern air force's arsenal. Besides transporting personnel, the Hercules may also take up goods and supplies for resupplement missions during peace and war times. In any case, mid-air ejection of goods and cargo via the tailgate always entails high risk for the C-130H and securing the airdropped goods' integrity is of topmost priority. Upsweep forces and recirculation zones pose as hazardous environmental factors as they may cause extraction parachutes and goods to be blown off track or become damaged in the extraction process.

There are several means of goods and supplies ejection system which are applicable to the C-130 and they are easily classified based on the altitude at which the airdrops are performed. This paper focuses on the Standard Airdrop Method designed to drop personnel and materials at an altitude that evades conventional radar systems.

The use of CFD is instrumental in this study as it attempts to predict the airflow situation and vorticity present at the mouth of the cargo bay. A 1:16 fuselage model will be first used to determine the DDES-SST benefits over DDES-SARC's. To save on computational resources then, simulations with the DDES-SST will be performed on a full-sized half axis-symmetric C-130H aircraft model.

Results present a three-dimensional view of strong unsteady vortices of magnitude 120 rotations/s evolving when the tailgate is lowered. Parachutes released in this area will have detrimental effect on the airdrop extraction from the cargo bay. A zone of zero-upsweep is measured from the lowered tailgate in hope to provide a safe distance for goods ejection. This distance is 3.85 m and 7.29 m at 0° and 5° of angle of attacks respectively. Height of this zone is also found to account for parachute diameters. With these findings, it is hopeful that they will be useful in the future designs of airdrop protocols.

Acknowledgements

I would like to express his heartfelt thanks to the people who made this thesis possible.

To Dr. Eddie Ng, a teacher, mentor and friend who spent countless hours patiently encouraging, guiding and providing me with advice throughout this two-year Master programme. Without his cordial and approachable nature, this stint would not have been as enjoyable as it had been.

I would like to thank Mr. Eric Yap from the Fluids Laboratory who has been faithfully supporting the project in the administration. It was his promptness and initiative that tremendously helped pushed for the software and computer resources to be delivered on time.

Special gratitude goes to Mr. Edwin Tan from HPCC who was instrumental in the initial stages of this project. It was a difficult and daunting six-month period when he was trying to install and create a suitable environment for the *Cobalt* software bundle which was user-friendly for me, but he managed to work everything out in the end. He was also always available to provide technical support during that period, pushing aside other important work that he had on hand in order to assist me in troubleshooting problems arising from *Cobalt's* licensing and running on the HPC. Many thanks to Mr. Leong Hon Wai, Mr.

Wong Foo Lam and the rest of the HPCC customer support team who were always quick in replying to queries and dealing with user-related problems with the HPC.

I wish to express my appreciation to Dr. Keith Bergeron, who provided this project with the C-130H fuselage model and grid, and to Dr. Bob Tomaro with his unceasing patience to instruct me on the proper usage in the *Cobalt* suite.

Finally, I wish to thank to the people at ALD, Ms. Chiu Wen Shan, ME3 Zhang Weisheng and ME3 Kevin Lim, because without their funding and opportunity given to me in this project, this Master programme would not have been possible. I am also very appreciative of the time they have taken out of their busy schedule to meet Dr. Eddie and I to understand the situations and difficulties that we faced in the course of the programme.

Table of Contents

Abstract	i
Acknowledgements	iii
List of Figures	viii
List of Tables	xi
Nomenclature	xii
Chapter 1 Introduction	1
1.1 Background	1
1.2 Aim	8
1.3 Objectives	8
1.4 Order of Presentation	9
Chapter 2 Literature Review	11
2.1 RANS Turbulence Modeling	11
2.1.1 Spalart-Allmaras Turbulence Model	12
2.1.2 Spalart-Shur Rotation and Curvature Correction	15
2.1.3 Menter's $k-\omega$ Shear Stress Transport Turbulence Model	15
2.2 Large-Eddy Simulation	19

2.3	Detached-Eddy Simulation	25
2.3.1	Derivation of DES	26
2.3.2	Application of DES	29
2.3.3	DES Issues	31
2.4	Delayed Detached-Eddy Simulation	32
2.4.1	Derivation of DDES-SST	33
2.4.2	Derivation of DDES-SA	33
2.5	Previous Work with C-130H	35
Chapter 3	Numerical Methods and Setup	43
3.1	Phase I – 1:16 C-130H Fuselage Model	43
3.1.1	Flow Domain and Grid	43
3.1.2	Numerical Settings	48
3.2	Phase II – Full-sized Half-symmetric C-130H Model	49
3.2.1	Flow Domain and Grid	50
3.2.2	Numerical Settings	55
Chapter 4	Results and Discussion	58
4.1	Phase I Results	58

4.2	Phase II Results	67
Chapter 5	Conclusion	88

References

List of Figures

Figure

- 1 Standard Airdrop Method sequence
- 2 Low altitude parachute extraction method sequence
- 3 Dimensions of the Phase I flow domain in which the C-130H fuselage sits in
- 4 Phase I's C-130H fuselage model
- 5 Mesh of the flow domain on the symmetry plane
- 6 A close-up view of the empennage end mesh
- 7 Planform view of the C-130H's head's surface mesh
- 8 Planform view of the C-130H's empennage
- 9 Phase II's C-130H in its computational domain
- 10 Dimensions of the computational domain from the back view
- 11 Double-domain Phase II mesh
- 12 Third domain in Phase II's triple-domain mesh
- 13 Streamlines of airflow around Phase I's C-130H fuselage
- 14 Graphs of lift and drag coefficients with varying AOA from -5° to 5°
- 15 Iso-surface of x-vorticity with magnitude of 750 revolutions/s evolved from the C-130H fuselage at 0 AOA. DDES-SARC was used as the turbulence model.
- 16 Iso-surface of x-vorticity with magnitude of 750 revolutions/s at 0° AOA with the DDES-SST turbulence model
- 17(a) Iso-surface plot comparisons between DDES-SARC and DDES-SST at 5°
- 17(b) Iso-surface plot comparisons between DDES-SARC and DDES-SST at -5°

- 18 Comparison of 100 rotations/s iso-surfaces at 40 m/s with the double- and triple-domain meshes
- 19 Lift and drag coefficient graphs of the C-130H with main wings at 40 m/s
- 20 Contour lines of velocity magnitude on the C-130H fuselage in Phase I
- 21 Vector plot of the downwash aft under the empennage. $AOA = 0^\circ$
- 22 Contour lines by velocity magnitude on the C-130H in Phase II. $AOA = 0^\circ$
- 23 Vector plot of the downwash aft under the empennage. $AOA = 5^\circ$
- 24 Contour lines by velocity magnitude on the C-130H in Phase II. $AOA = 5^\circ$
- 25(a) Vorticity iso-surface plots at 120 rotations/s at the mouth of the cargo bay with $AOAs$ of 0° .
- 25(b) Vorticity iso-surface plots at 120 rotations/s at the mouth of the cargo bay with $AOAs$ of 5°
- 26 Streamlines on the underside of the aircraft demonstrating the change in airflow direction
- 27 Lift and drag coefficient graphs comparing an opened and closed cargo bay
- 28(a) Iso -surfaces of vorticity magnitude 120 rotations/s at $AOA = 0^\circ$
- 28(b) Iso -surfaces of vorticity magnitude 120 rotations/s at $AOA = 5^\circ$
- 29(a) Velocity contours of the vortices evolved at the tailgate's edge. $AOA = 0^\circ$
- 29(b) Vectors of the vortices evolved at the tailgate's edge. $AOA = 0^\circ$
- 30(a) Velocity contours of the vortices evolved at the tailgate's edge. $AOA = 5^\circ$
- 30(b) Vectors of the vortices evolved at the tailgate's edge. $AOA = 5^\circ$
- 31 Measurement of the distance of zero-upswEEP from the tailboard's edge. $AOA = 0^\circ$
- 32 Measurement of the height of zero-upswEEP window. $AOA = 0^\circ$
- 33 Measurement of the distance of zero-upswEEP from the tailboard's edge

$$\text{AOA} = 5^\circ$$

- 34 Measurement of the height of zero-upsweep window. $\text{AOA} = 5^\circ$

List of Tables

Table

- 1(a) Comparison of C_L of Phase I.
- 1(b) Comparison of C_D of Phase I.

Nomenclature

a	=	speed of sound in air in m/s
δ_{ij}	=	Kronecker delta
Δ	=	distance between cell center in concern with neighbor cell center
Δ_{DES}	=	largest distance between current cell center and neighbors'
D_0	=	nominal diameter in ft or m
D'_p	=	projected diameter in ft or m
d	=	distance to the nearest wall
ε	=	turbulent dissipation
E_{jmn}	=	Levi-Civita tensor
G	=	filtering function
γ	=	ratio of specific heats
k	=	turbulent kinetic energy
l	=	length scale
M	=	Mach number
ν	=	kinematic viscosity in kg/(ms)
ν_m	=	molecular viscosity
ν_t	=	turbulent eddy viscosity
$\overline{\Omega}$	=	vorticity magnitude in rotation/s
Ω_{ij}	=	component of vorticity

ω	=	specific dissipation
$\tilde{\phi}$	=	filtered function
ϕ	=	unfiltered function
p	=	fluid pressure in Pa
p_G	=	gauge pressure in Pa
ψ	=	cutoff width
ρ	=	fluid density in kg/m ³
S	=	strain rate tensor
S_{ij}	=	strain rate tensor component
$\bar{S}_{ij}$	=	mean strain rate tensor
σ	=	Prandtl number
T	=	temperature in K
τ_{ij}	=	Reynolds stresses
u	=	fluid velocity in m/s
u_i, u_j	=	components of fluid velocity
u'	=	fluid velocity fluctuation
x	=	spatial tensor
x'	=	spatial tensor fluctuation
x_i, x_j	=	component of spatial tensor

1 INTRODUCTION

1.1 Background

The Lockheed C-130 Hercules is arguably one of the most versatile fixed-wing transport aircraft available in any modern air force's arsenal. The concept of the C-130 was borne in the early 1950s when the C-46 Commando, C-119 Flying Boxcar and C-47 Skytrain became obsolete after the Korean War^[1]. The United States Air Force (USAF) needed an aircraft which could transport at least 92 passengers, 72 troopers or 64 paratroopers over a distance of 1100 nautical miles, perform take-offs on unprepared runways (short and even dirt runways) and maintain altitude and tactical operations even with a failed engine. Lockheed immediately presented the USAF with the YC-130 prototype in 1954 and meeting the requirements.

Since then, over 40 variants of the C-130, which include assault types, have been developed. The capabilities of the aircraft now range from aerial fire-fighting, tactical airlifts and even acting as bombing. Some of the popular non-combatant variants include C-130A, B, D, E, G and H, which among them A, B and D have been decommissioned from service.

The most widely used and exported variant, together with its sub-variants, is currently the C-130H, or Hercules, first introduced in 1974 as an improvement to the E variant. Hercules is powered by four Allison T56-A-15 turboprops, has a length of 29.3 m, height of 11.9 m and a wingspan of 39.7 m^[2]. The cargo compartment is 12.31 m long, 3.12 m wide and 2.74

m high. Besides transporting passengers, the Hercules may also take up to any configuration of 6 pallets of goods, 4 jeeps and 4 trailers in any configuration, as long as it adheres to the maximum allowable payload of 19090kg. As a result, the C-130H is also known informally as the “airdrop work horse”^[3].

A cargo ramp, also known as the tailgate or tailboard, is 3.12 m long by 3.02 m wide and located aft of the aircraft. The ramp provides the primary loading and unloading access of goods into the cargo bay and is also the main platform on which tactical mid-flight ejections are performed.

There are several means of goods and supplies ejection system which are applicable to the C-130 and they are easily classified based on the altitude at which the airdrops are performed.

The Standard Airdrop Method (SAM)^[3] as seen in Figure 1 is designed to drop personnel and material at 50 to 1500 feet at aircraft velocities ranging from 120 to 150 knots so as to avoid enemy radar detection and anti-aircraft artillery (AAA) counter-action. As the aircraft nears the drop site, the pilot issues the command and the drop procedure is as follows: the extraction parachute is deployed to pull the cargo and main parachute(s) out from the cargo bay via the tailgate. The main parachute(s) are chief in stabilizing and ensuring a proper descent of the cargo at a descent speed, ideally at 20 to 30 fps or 6 to 9 m/s. Finally the parachute(s) disconnect fully when the cargo has safely landed and is ready for recovery.

Figure 1: Standard Airdrop Method sequence. Pictorial taken from Reference [3].

The Low Altitude Parachute Extraction Method (LAPES) is also another common way of delivering materials and goods. The method is displayed in Figure 2. Currently the C-130 is the only aircraft approved and able to perform LAPES. The aircraft will approach the drop zone at a low altitude of approximately 5 ft or 1.5 m and at a low velocity. A drogue parachute is deployed to extract the main parachute(s). The drag force captured by the main parachute(s) tows the goods out the back of the aircraft and decelerate them to a halt. The C-130 then makes a tactical ascent to leave the drop zone.

Figure 2: Low altitude parachute extraction method sequence. Pictorial taken from Reference [3].

Flying at such low altitudes allows the C-130 to be below the radar detection range and unlike the standard method, LAPES also prevents AAA from attacking both aircraft and materials dropped. Unfortunately, to perform such a procedure requires good piloting skills as well as a relatively large area of flat terrain for the aircraft to approach and for parachutes to decelerate.

Recently, newer technologies have improved the airdrop systems by leaps and bounds, enabling the C-130 to even perform tactical drops with greater precision at high altitudes. The success (and failure) of such depend largely on the equipment used to control the deceleration and stability of the cargo. Parafoils promote gliding and are easier to control in

terms of its lateral movements. Furthermore with finer gadgets attached to parafoil, they enable it to brave drafts and gusts without losing its direction to its drop zone.

In any case, mid-air ejection of goods and cargo via the tailgate always entails high risks for the C-130H. Securing the airdropped goods' integrity is of topmost priority. As soon they are ejected, the parachute systems begin the extraction and deceleration processes. However parachutes may be caught in the turbulence generated aft of the aircraft due to zones of massive flow separations and recirculation. The upsweep also may drag parachutes, and inevitably the cargo, up towards the empennage. This is especially the case when the parachutes' suspension lines and the riser connecting the goods to suspension lines are too long.

This upsweep draft may have a force large enough to propel the parachute, together with its cargo, upwards, veering it off course from the drop zone. In more destructive scenarios, both aircraft and goods might be damaged as the parachute(s) are brought close the tail end, while the turbulent vortices perpetually slam the goods against the aircraft body or tail. With such potentially undesirable circumstances to be considered, determining the locations and propagation of vortices as well as their magnitudes have especially significant importance to military engineers because this would greatly assist resupplement of vehicles, ammunition and combatant supplies, and ensuring improved rates of success in missions.

As part of the scenario design process, computational fluid dynamics (CFD) was employed as a step in the analysis for the C-130H cargo extraction system. There are several advantages with using CFD:

For one, CFD is a more cost-effective method as compared to repeated wind tunnel and full-scale live testings. It allows engineers to easily set flight parameters and change aircraft designs quickly so that they may gather a “feel” of what the aircraft will be subjected to in the actual case. In some cases where controlled or dangerous environments are required and difficult to achieve in real experiments, CFD evidently becomes an asset. Furthermore, the time and labour cost of producing a computer-aided design (CAD) of an aircraft of any degree of complexity is less than fabricating a scaled wind tunnel model and far shorter than manufacturing a full-sized test aircraft. Solving the fluid dynamics may also be hastened by dividing up the flow domain and running parallel solution processes simultaneously.

With CFD, information such as the wall-shear stresses along a body, vorticity magnitudes, turbulent kinetic energy and the like, which are usually difficult or impossible to obtain experimentally, may now be reflected as part of the solution report. Besides that, multiple data may be mined from just a single simulation as opposed to having to run several live tests in order to gather the same sets of data.

Therefore CFD has become an integral part of the design process in many research areas, especially in the aviation sector. Of course, CFD is still at best a supplement to actual live

testing, which includes wind and water tunnel analyses. Once CFD has achieved its purpose of attaining that “feel” for the situation, the next step would be to run a real case to confirm the validity of the results obtained thus far. In this project, CFD is used to visualize more vividly the vortices which are spawned near the tailgate and that might invite hazards to follow.

The project is part of the Republic of Singapore Air Force’s (RSAF) and the Defence Science and Technology Agency’s (DSTA) efforts to improve upon and customize the current knowledge of precision airdrop systems for local use. There has already been several research papers published and presentations made pertaining to C-130H airdrops especially from the Airflow Influence on Airdrop (AIA) project. The AIA project was a joint collaboration which started in July 2002 between the United States of America, the Republic of France, the Federal Republic of Germany and the United Kingdom of Great Britain and Northern Ireland^[4].

One of the aims of the AIA project was to look into the airflow around the C-130H and K aircrafts, cargo pallets and parachutes and the dynamics between them when they are coupled together during an airdrop mission. By meeting this aim, it meant that the team would have been able to seek measures to improve parachutists, load and aircraft safety, while at the same time increase the airdrop efficiencies of both the C-130H and K.

1.2 Scope

The aim of this project is to determine the clearance distance of which goods may be ejected from the cargo bay with as little disturbance from the vortices generated at the mouth of the bay. This would include measuring the sizes and magnitudes of the larger vortex bubbles evolved. The results obtained might be beneficial for the RSAF-DSTA as they re-look and design protocols for safer C-130 airdrops and possibly to improve the ejection systems such that they suffer less impact from the upsweep.

This CFD project will also serve as a springboard by which new ideas pertaining to airdrop efficiency and accuracy to the drop zone could be further implemented. The information to these ideas however will not be disclosed in this report as they are classified with confidentiality.

1.3 Objectives

The project has two phases which utilizes two CFD solvers to resolve flows over the C-130H. Phase 1 is essentially a familiarization to the *Cobalt* solver suite and a validation experiment to compare its outcome with past simulations done. It also studies of the effects on airflow near the empennage of a simplified C-130H fuselage with its tailgate closed. As it is expected that the flight Reynolds number will be high, the Delayed Detached-Eddy

Simulation with Menter's Shear Stress Transport (DDES-SST) turbulence model will be used.

The second phase is the more critical part of the project as it looks at a full C-130H with wings and its tailgate lowered at flight velocities of 40 and 72 m/s with angle of attacks (AOA) ranging from -10° to 10° . In this phase, ANSYS FLUENT is utilized as the solver. Massively detached flows are expected at the mouth of the cargo bay, hence vortex magnitudes will be measured while the location and extent of the recirculation zone have to be analyzed. A comparison of turbulence models will also be performed as part of the analysis. This model possesses horizontal wings, thus this phase will look into the possible effects of wings on the empennage and the mouth of the cargo bay.

1.4 Order of Presentation

The report starts with the first chapter presenting an introduction to the C-130H aircraft, some airdrop procedures that are currently in place and the motive behind using CFD as the mode of preliminary testing to improve designs of procedures. Besides that, the chapter provides the scope and aims for the paper.

Chapter 2 will cover the history and progress of CFD and review past works with CFD on the C-130H in particular the upsweep forces experienced aft of the aircraft and its effects on the extraction process. Chapter 3 will describe the numerical setup and methods for this report's simulations with the *Cobalt* and ANSYS FLUENT solvers.

A detailed discussion of the results will be presented in the fifth chapter, explaining the differences between two delayed detached-eddy simulation turbulence numerical schemes. It will also highlight the aerodynamics of the airflow at the mouth of the cargo bay and propose a safe distance from the tailgate at which a parachute should be inflated in order to avoid the problematic situation near the cargo bay.

Finally, the report will close with a few concluding remarks in Chapter 6 as well as suggest future ideas that may be beneficial and interesting to implement in the aspect of CFD for airdrop protocols.

2 LITERATURE REVIEW

This segment will run through a brief history of and the motivation for the development of the Delayed Detached-Eddy Simulation (DDES).

It is expected that in normal flight conditions, the C-130H will be in the range of high Reynolds numbers. At such situations, a lot of unsteadiness will be generated in the flow around the aircraft due to the high ratio between fluid inertia to viscosity. Therefore it is expected that with a lowered cargo ramp during extraction, a lot of turbulence is generated in its near wake.

Turbulence modeling in conventional CFD was by means of applying a Reynolds-averaged Navier-Stokes (RANS) scheme to the calculations. Over the years, there have been many RANS equations formulated for various purposes. Some of the more common models are the single transport equation Spalart-Allmaras (SA), double-equations $k-\varepsilon$ and $k-\omega$, and even the more complex Reynolds stress equation model (RSM).

2.1 RANS Turbulence Modeling

RANS models are named as so because their focus is on the derivation of mean flow properties in turbulence, in particular the Reynolds shear stresses. These properties are extracted by first time-averaging the Navier-Stokes equations before applying any numerical method to solve for turbulent flow. In most engineering applications, resolving

these mean flow properties and getting an approximate overview of the flow situation is all that is required. At the same time, most RANS models are simple to implement and may be used for a wide range of scenarios, therefore it still dominates the CFD arena where turbulence prediction is concerned.

RANS modeling is built upon the Boussinesq hypothesis which states that the Reynolds stresses are proportionally related to the average rate of deformation of a fluid particle, or

$$\tau_{ij} = \nu_t S_{ij} - \frac{2}{3} \rho k \delta_{ij}, \text{ where } k = \frac{1}{2} u'^2 \text{ and } \delta_{ij} = \begin{cases} 1, & i = j \\ 0, & i \neq j \end{cases}. \text{ The Boussinesq hypothesis is}$$

simply just a re-iteration of Newton's law of viscosity for an incompressible fluid.

2.1.1 Spalart-Allmaras Turbulence Model

In the aerospace sector, the Spalart-Allmaras single transport equation turbulence model (SA)^[5] is one of the most commonly used in CFD research. It was formulated by Spalart and Allmaras in 1992 as a reaction to the single equation models already in existence which were algebraic and, at best, boundary layer models. Even so, these single equation models could only solve for boundary layers with the assumption that they were a thin single layer. However, in real situations, this was not always the case. Boundary layer flows can be detached and possess many shear layers as well. Furthermore, there was no continuity between the solutions derived at the boundary layer around the aircraft and the wake region where eddy viscosity was concerned.

These above reasons were the drive for a better single equation model to be worked out. The SA was to be able to improve the accuracy of solutions and especially with unstructured grids, which were becoming more popular in the day and were cumbersome for the algebraic models to compute. At the same time, Spalart and Allmaras wanted a model that was computationally cheaper in terms of resolution of grid cells at near wall regions and also simpler to use than the well established two-equation models.

Since the SA model was to maintain continuity to the eddy viscosity from the aircraft region to the wake, the working quantity naturally was ν_t . They also introduced a constructed working variable $\tilde{\nu}$, which will be equal to ν_t in all regions of the flow except at the viscous layers. At the wall boundary, $\nu_t = 0$. This new quantity allowed for less grid resolution than that required of the algebraic models because it is more easily resolved than other aspects of the flow velocity.

His interpretation of the Reynolds stresses had the turbulent kinetic energy term removed. He reasoned that it was just an approximate figure to balance the stress tensor's diagonal elements and they did not affect thin shear layers much in any case. Therefore the Reynolds stresses was reduced to $\tau_{ij} = 2\nu_t S_{ij}$, which would ultimately help reduce a two-equation model into a single-equation. They were still able to attain a rough estimate of the turbulent kinetic energy by assuming that k is proportional to the Reynolds stresses, or $k \propto \nu_t S_{ij}$.

The final equation constructed was Lagrangian in nature and possessed a production, diffusion and destruction term:

$$\frac{D\tilde{v}}{Dt} = c_{b1}\tilde{S}\tilde{v} + \frac{1}{\sigma}\left\{\nabla \cdot [(v_m + \tilde{v})\nabla\tilde{v}] + c_{b2}(\nabla\tilde{v})^2\right\} - c_{w1}f_w\left(\frac{\tilde{v}}{d}\right)^2 \quad (1)$$

with the constituents to the equation being

$$\begin{aligned} \chi &= \frac{\tilde{v}}{v_m} & v_t &= \tilde{v}f_{v1} & f_{v1} &= \frac{\chi^3}{\chi^3 + c_{v1}^3} \\ \tilde{S} &\equiv S + \frac{\tilde{v}}{\kappa^2 d^2} f_{v2} & f_{v2} &= 1 - \frac{\chi}{1 - \chi f_{v1}} & f_w &= g \left(\frac{1 + c_{w3}^6}{g^6 + c_{w3}^6} \right)^{1/6} \\ g &= r + c_{w2}(r^6 - r) & r &\equiv \frac{v_t}{S\kappa^2 d^2} & c_{w1} &= \frac{c_{b1}}{\kappa^2} + \frac{1 + c_{b2}}{\sigma} \end{aligned}$$

The constants proposed were as follows:

$$\begin{aligned} \sigma &= \frac{2}{3} & c_{b1} &= 0.1355 & c_{b2} &= 0.622 \\ \kappa &= 0.41 & c_{v1} &= 7.1 & c_{w2} &= 0.3 \\ c_{w3} &= 2 \end{aligned}$$

2.1.2 Spalart-Shur Rotation and Curvature Correction

The Spalart-Shur rotation and curvature correction to the SA equation (SARC) was formulated because the SA was inadequate in predicting turbulent shear flows around

convex bends^[6]. The modification is simple because it only requires a rotation function f_{r1} to be multiplied to the production term of the SA equation, that is $P \equiv c_{b1} \tilde{S} \tilde{\nu}$. The function takes the form of

$$f_{r1}(r^*, \tilde{r}) = (1 + c_{r1}) \frac{2r^*}{1 + r^*} [1 - c_{r3} \tan^{-1}(c_{r2} \tilde{r})] - c_{r1} \quad (2)$$

with a series of supporting variables defined as

$$\begin{aligned} r^* &= \frac{S}{\bar{\Omega}} & \tilde{r} &= 2\Omega_{ik} S_{ik} \left[\frac{DS_{ij}}{Dt} + (E_{imn} S_{jn} + E_{jmn} S_{in}) \Omega_m \right] / D^4 & S^2 &= 2S_{ij} S_{ij} \\ \bar{\Omega}^2 &= 2\Omega_{ij} \Omega_{ij} & \Omega_{ij} &= \frac{1}{2} \left[\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) + 2E_{mji} \Omega_m \right] & D^2 &= \frac{1}{2} (S^2 + \bar{\Omega}^2) \end{aligned}$$

2.1.3 Menter's k - ω Shear Stress Transport Turbulence Model

Another popular turbulence model that has been formulated at around the same time for aeronautical purposes was the Menter's Shear Stress Transport (SST) model. The older k - ε and k - ω models have been popular thus far, however their solutions were mutually exclusive. That is to say, the k - ε was, and still is, good in solving for turbulent flow variables in the freestream but inadequate at near-wall boundary areas where strong pressure gradients are present, whereas the Wilcox k - ω works in the reverse. This is the case because of the choice of length scale used for each model. The length scale of the k - ε makes use of the turbulent energy dissipation to derive $l = \sqrt{k^3}/\varepsilon$ and the k - ω , $l = \sqrt{k}/\omega$,

where $\omega = \varepsilon/k$ is the turbulent frequency^[8]. These observations were made by Menter in his review of four popular turbulence models^[9].

It is possible to have a two-equation model which worked much like the Wilcox model with far less dependency on the freestream values which were ambiguous. In 1993, Menter proposed two models which merged both the k - ε and k - ω , creating hybrids which were effective in both near-wall boundary layers and freestream solutions^[9]. One of which is the SST model, which he assured to be only slightly more complicated and would take a little more computational resources than the original k - ω of Wilcox. Other than that, its attributes would fare far better than any two-equation model of the time. In fact, the SST is currently regarded as one of the most accurate two-equation models available for aerodynamic research.

His SST equations thus would take the form

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \tilde{P}_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_i} \left[(v_m + \sigma_k v_t) \frac{\partial k}{\partial x_i} \right] \quad (3a)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_i \omega)}{\partial x_i} = \alpha \rho S^2 + \frac{\partial}{\partial x_i} \left[(v_m + \sigma_\omega v_t) \frac{\partial \omega}{\partial x_i} \right] + 2(1 + F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \quad (3b)$$

The length scale which Menter took was $l_{SST} = \sqrt{k}/(\beta^* \omega)$. The function F_1 is called the first blending function to merge both k - ε and k - ω equations. It is defined as

$$F_1 = \tanh(\arg_1^4) \quad (4)$$

with

$$\arg_1 = \min \left[\max \left(\frac{\sqrt{k}}{0.09\omega d}, \frac{500\nu_m}{d^2\omega} \right), \frac{4\rho\sigma_{\omega 2}k}{CD_{k\omega}d^2} \right] \quad (4a)$$

and

$$CD_{k\omega} = \max \left(2\rho\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right) \quad (4b)$$

This argument basically allows the SST equation to switch from k - ω in the boundary layer to k - ε far from the wall because $d \rightarrow \infty$, causing $F_1 \rightarrow 0$.

Two-equation models tend to produce higher turbulent shear-stress because the ratio of the production to dissipation terms may exceed 1 in regions of adverse pressure gradients. In order to fulfill the constraints of Bradshaw's assumption—boundary layer shear-stress is proportional to the turbulent kinetic energy or $\tau = \rho a_1 k$, a_1 being a constant—Menter deviated from the conventional definition of the eddy-viscosity and re-introduced it in his SST equations as

$$\nu_t = \frac{a_1 k}{\max(a_1\omega, SF_2)} \quad (5)$$

where F_2 is the second blending function:

$$F_2 = \tanh(\arg_2^2) \quad (6)$$

$$\arg_2 = \max\left(\frac{2\sqrt{k}}{0.09\omega d}, \frac{500v_m}{d^2\omega}\right) \quad (6a)$$

This blending function now enables v_t to have its greatest value near the wall instead of the freestream region.

Lastly, Equation (3a) also provides a term called the production limiter in regions where $\omega = 0$ and $v_t \ll v_m$ such that the production term escalates into infinity with even small disturbances of the rate of strain. This usually occurs at stagnation regions. Therefore, the production limiter would stop any of these unwanted turbulence by

$$\tilde{P}_k = \min(P_k, 10 \cdot \beta^* \rho k \omega) \quad (7)$$

while

$$P_k = v_t \frac{\partial u_i}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (7a)$$

The rest of the constants in the SST equations were taken from both the k - ϵ and k - ω :

$$\begin{aligned} \alpha &= \alpha_1 F_1 + \alpha_2 (1 - F_1) & \alpha_1 &= \frac{5}{9} & \alpha_2 &= 0.44 \\ \beta^* &= 0.09 & \beta_1 &= \frac{3}{40} & \beta_2 &= 0.0826 \\ \sigma_{k1} &= 0.85 & \sigma_{k2} &= 1 & \sigma_{\omega1} &= 0.5 \\ \sigma_{\omega2} &= 0.856 \end{aligned}$$

Both the SA and Menter's SST models are among the most popular for aerospace CFD research due to their low cost of computational resources and their ease of usage. As mentioned, many applications just require time-averaged or mean flow analyses to get the job done, therefore RANS still remains by and large the dominant form of turbulent modeling in the industry.

2.2 Large Eddy Simulation

In recent years though, there has been a rising trend to observe the time-dependent nature of detached flows where eddying vortices may grow large. RANS tend to function best in areas of attached flow where Reynolds numbers are relatively low and where eddies are isotropic due to the energy cascade. As Reynolds numbers increase, the size of eddies also increase and flows become detached. Beyond a size limit, they turn anisotropic because of the shear stress interactions between large eddies. Anisotropic eddies are present in all massively detached flows, which are geometry-dependent, and to use RANS models in such situations warrants inaccuracies to arise in solutions.

A primitive set of large eddy simulation (LES) equations was presented by Smagorinsky in 1963 as a means to predict global weather patterns by simulating the unsteady circulation of the atmosphere^[11]. Weather patterns are transient in nature and may have periods of high turbulence due to atmosphere heating and subsequent dissipation of heat energy, not to mention the cyclonic phenomenon which occurs when cold and hot fronts mix.

To verify the LES method, Deardorff performed a series of computational experiments of atmospheric turbulence prediction by means of testing the effects of planetary heating on the Earth's boundary layer and subsequently the boundary layer's contribution to weather changes^{[12][13][14]}.

The LES works on the basis of spatial-averaging or spatial-filtering as opposed to time-averaging. It acts as a low-pass filter, capturing only the effects of the large eddies while letting the small eddies pass through the filter. Large eddies, according to Ferziger, cannot and will not in any case be modeled^[15] whereas it is possible to readily and adequately model the small eddies. By producing an averaging function which distinguishes the small eddies, the remaining eddies left in the flow are substantially computed and considered to be large eddies.

Since atmospheric conditions are less affected by near-earth boundary layers (large eddies do not exist in boundary layers), the LES was an adequate tool for weather prediction. In the same way, when applied to aerodynamic purposes, the LES would be able to perform better analyses on regions of massively separated flows that are mostly detached away from boundary layers.

The filtered function is given by

$$\tilde{\phi}(x,t) = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} G(x,x',\psi) \phi(x',t) dx'_1 dx'_2 dx'_3 \quad (8)$$

The filtering process is done via the limiting factor ψ where in three-dimensional space, $\psi = \sqrt[3]{\Delta x \Delta y \Delta z}$. ψ basically limits the size of the eddies which may pass through the filter.

G may take many forms according to the needs of the simulation, but the three most common functions are the

box filter,

$$G = \begin{cases} 1/\psi^3 & |x - x'| \leq \psi/2 \\ 0 & \text{otherwise} \end{cases} \quad (8a)$$

Gaussian filter,

$$G = \sqrt{\left(\frac{6}{\pi\psi^2}\right)^3} \exp\left(\frac{-6|x - x'|^2}{\psi^2}\right) \quad (8b)$$

and spectral cutoff

$$G = \prod_{i=1}^3 \frac{\sin[(x_i - x'_i)/\psi]}{(x_i - x'_i)/\psi} \quad (8c)$$

which is essentially a filter function that is manipulated in Fourier space.

The box filter was an easy and hence popular filter to use by the early pioneers of LES. Among them to apply LES to an engineering problem (channel flow study) rather than a

weather scenario was Deardorff^[16]. He was the first to deviate from the norm, who were then adamant that turbulent modeling was only acceptable by RANS. With all that said though, the filtering function was meant to just sieve out the large eddies in solving the Navier-Stokes equations, from the incompressible form of the Navier-Stokes equations

$$\frac{\partial u_i}{\partial t} + \frac{\partial(u_i u_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \nabla^2 u_i \quad (9)$$

to its filtered averaged form

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial(\overline{u_i u_j})}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \nabla^2 \bar{u}_i \quad (9a)$$

This gave rise to a set of mean flow variables derived from the mean turbulent flow. Expanding the new product velocity term $\overline{u_i u_j}$ revealed the subgrid scale velocity fluctuation quantity u' :

$$\overline{u_i u_j} = \overline{\bar{u}_i \bar{u}_j} + \overline{\bar{u}_i u'_j} + \overline{u'_i \bar{u}_j} + \overline{u'_i u'_j} \quad (10)$$

The Leonard stresses, $\overline{\bar{u}_i \bar{u}_j}$, are borne out of a second filtering process of $\bar{u}_i \bar{u}_j$ and this may be approximated using the Taylor series. The cross-stresses, $\overline{\bar{u}_i u'_j} + \overline{u'_i \bar{u}_j}$, are formed by the subgrid scale fluctuations mixing with the mean flow. Lastly, the Reynolds stresses $\overline{u'_i u'_j}$, evolved because of the interactions between the small and large eddies, which results in the energy cascade.

It is essential to model all these fluctuation terms, especially the Reynolds stresses. Since the energy cascade is evident in high Reynolds numbers and massively separated flows, eddy viscosity becomes a good representation for these subgrid scale terms.

$$\overline{u'_i u'_j} \approx \nu_t \bar{S}_{ij}, \quad \bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \quad (11)$$

Smagorinsky thus proposed in his formulation of LES that

$$\nu_t = (c_{SGS} \psi)^2 \sqrt{\bar{S}_{ij} \bar{S}_{ij}} \quad (12)$$

c_{SGS} is the subgrid scale constant. Lilly suggested that the constant value should reflect the energy dissipation of isotropic small eddies^[17]. He produced a range where $c_{SGS} = [0.17, 0.21]$, however this range was not universal. Several other scientists including Rogallo and Moin have tried to fine tune the c_{SGS} to suit more specific applications^[18]. They remarked that a c_{SGS} with a value between 0.19 and 0.24 works well for external flows while for internal flows in a channel, $c_{SGS} = 0.1$ is more apt.

Putting all these terms back into the filtered equations of the Navier-Stokes enabled the classical Smagorinsky-Lilly LES model, where Lilly's contribution was in the description of the eddy viscosity modeling. There have been new advances in the LES method with higher level subgrid scale models since then, and more so in the recent decade as computational resources become cheaper and more powerful, making LES a very robust tool in CFD^[19-23].

Deardorff's application of LES on channel flows has already been mentioned as a proper validation of the technique to CFD^[16]. Several others have also further contributed to testing the model on a variety of applications, from simple to complex geometries and situations^[24-27].

Constantinescu *et al.* explored the LES capabilities on the basic CFD case of flow over a sphere^[24]. He found that it was able to accurately predict the location of flow separation from the boundary layer into turbulent flow. It was able to capture clearly the turbulent structures and flow properties that formed in the wake and could collect favorable data of drag, Strouhal numbers and so on with respect to experimental results. The test also showed that LES was unable to give a good description of the wake turbulence just immediately aft of the sphere.

Moin then reviewed a series of more complex simulations, all proving the worthiness of the LES method^[25]. One of them performed was on a Pratt & Whitney gas-turbine engine by Mahesh *et al.* at Stanford University^[26,27].

The grid was composed of unstructured tetrahedral and hexahedral cells. Since the conception of LES, unstructured meshes have rarely been utilized as an option for the turbulence model. Mahesh subdivided the turbine into multiple blocks for different turbulence schemes: RANS for turbine and compressor blocks and LES for the rest of the combustor sections. According to the need for computational accuracy each block was treated with either the hexahedral or tetrahedral cells. The conclusion to that experiment

was that LES could very adequately capture the swirls, mixing of fluid and the like in the combustor, which had very high Reynolds numbers of 10^9 , without any difficulty.

The reason for Mahesh splitting the domain into blocks for RANS-LES models implementation was that although LES was good in massively detached flows at high Reynolds numbers, near-wall areas and thin boundary layer flows were a boon for the method. To adequately use LES as a general tool for all kinds of situations would sometimes mean that more meticulous work had to put into generating grids that are extremely fine near boundaries. This is because the cutoff width that is associated with LES would have to change at every so few intervals. That way, the computed subgrid scale stresses would not be in the same order as the truncation error of the solution. Such has been done before but the process is tedious and may be computationally cumbersome^{[28],[29]}.

These reasons make the application of LES for large projects like aircraft design analysis very costly and according to Spalart *et al.*, will continue to be so in the years to come^[30]. Therefore there has to be a more universal model which could perform the work RANS and LES at the same time.

2.3 Detached Eddy Simulation (DES)

The DES was the answer to the desire for a universal RANS-LES hybrid model designed by Spalart *et al.* for any situation of turbulence^[30]. As its name suggests, the DES primarily aims to solve for detached larger eddies in massively separated flows while still

maintaining an accurate portrayal of the attached smaller eddies which occur at thin shear layers.

2.3.1 Derivation of DES

The derivation of the DES was originally drawn out using the SA equation by placing a limiter for the variable d , transforming the quantity into

$$\tilde{d} = \min(d, C_{DES} \Delta_{DES}) \quad (13)$$

The Δ_{DES} here is referred to as the largest distance between the current cell center of the cell in concern with those of the neighboring cells. This is especially useful for the finite volume method of CFD which makes use of the cell centers to contain the information of flow fluxes in and out of the control volume. In other methods and applications Δ_{DES} could also be indicative of the largest dimension of the three-dimensional cell or $\Delta_{DES} = \max(\delta x, \delta y, \delta z)^{[7],[31]}$. Nonetheless, the definition of Δ_{DES} remains to be compared to the length scale quantity.

Spalart discussed that the original DES constant should be set at $C_{DES} = 0.65$ in homogenous turbulence.

With the limiter in place, the DES then has the ability to choose between SA and LES. If $d < C_{DES} \Delta_{DES}$, the algorithm would select SA because the DES interprets the cell in

concern to be within near-wall or boundary layer area. This is on the assumption that cells in the boundary layer are very thin, creating large anisotropy as the dimension parallel to the boundary surface is very much longer than its sides. The reverse of this just activates the LES. This DES model is thus a simple, intuitive and very much automatic hybrid system which would be perfect for universal CFD use, especially where non-uniform grids are concerned.

The DES is not limited to just the SA equation. Strelets commented that as it was also possible that the DES be applied with Menter's SST's two equations in the stead of the RANS modeling^[32].

Recalling that the length scale of the SST is $l_{SST} = \sqrt{k}/(\beta^* \omega)$ and this is present in the dissipative term $D = \rho k \beta^* \omega$ in SST. With the DES implementation, it converts the length scale into

$$\tilde{l} = \min(l_{SST}, C_{DES} \Delta_{DES}) \quad (14)$$

and the dissipative term becomes

$$D_{DES} = \rho \beta^* k \omega F_{DES} \quad (15)$$

where $F_{DES} = \max\left(\frac{l_{SST}}{C_{DES} \Delta_{DES}}, 1\right)$.

The F_{DES} blending function is then used to create a new C_{DES} such that it takes in account of both $C_{DES}^{k-\omega}$ and $C_{DES}^{k-\varepsilon}$ from the two equations. The new C_{DES} is as follows:

$$C_{DES} = (1 - F_1) C_{DES}^{k-\varepsilon} + F_1 C_{DES}^{k-\omega} \quad (16)$$

whereby $C_{DES}^{k-\omega} = 0.78$ and $C_{DES}^{k-\varepsilon} = 0.61$.

Another advantage of the DES over the LES is that computationally it is a lot less expensive. That is to say at thin shear layers, the DES would take on the modeling requirements of RANS instead of LES. This idea however must be properly managed as Spalart commented in a subsequent report about the DES^[33].

It is incorrect to say that the DES allows a coarser grid than LES does. The fact that the DES is a hybrid system means it has to adhere to the grid requirements of both RANS and LES^[33]. Instead of incorporating a highly refined cell density in the boundary layer to resolve the isotropic small eddies as pure LES does, the DES is able to lower that resolution and still attain decent flow prediction because of the RANS capability. On the other hand, as the DES transits from RANS to LES beyond the logarithmic layer, the grid density must be of the same density as the LES in order to capture comprehensively the flow variables in this region.

2.3.2 Applications of DES

The DES, despite being a young and new turbulence concept, has been well received by many in the CFD arena, especially in the department of aeronautical research.

In the same paper done to validate the LES using the flow past a sphere, Constantinescu and Squires also performed a few DES studies to compare the results^[24]. Their conclusion was that the DES in general worked very similarly to the LES in terms of prediction of vortex shedding, Strouhal numbers and other flow properties. The exception to the DES was that it has the added advantage of being able to provide for the solutions at the near wall boundary.

They also found that should they increase the C_{DES} to 1.6 and even 2.0, a larger damping function was induced such that the energy of higher eddying wavenumbers was dissipated away. This was an effect similar to what is known as artificial viscosity or artificial dissipation. The fact is that the C_{DES} is associated with the eddy viscosity production. As C_{DES} increased, eddy viscosity increased as well, bringing about more inaccuracies.

Another CFD classical case on which DES was applied to was the flow past a cylinder by Travin *et al.*^[31]. He tested it with Reynolds numbers of up to 3.0×10^6 . It was noted that the turbulence model expressed a good three-dimensionality to the solution on a geometry which was only two-dimensional and the Strouhal numbers were within experimental range.

He commented too that the DES portrayed properties of the separated flow in the wake far better than the unsteady RANS (URANS) method which they also applied.

The DES has also been used to simulate a F-15E aircraft in flights of high angle of attack (AOA)^[34]. In his analysis, Forsythe *et al.* compared the SA derived DES, or DES-SA, with the SA model and the effects of coarse grids as opposed to finer grids on the DES-SA.

Application of the DES presented far better resolution of vortices and was much more accurate in lift, drag and moment than the SA. The SA tended to over-predict these flight coefficients, +7% for drag, +10% for lift and +8% for the moment. Even with a coarse grid, some of turbulent structures were still clearly visible along the wings with the DES whereas the SA could only vaguely demonstrate the presence of the wing-tip vortices. And as expected, the coarse grid would fare worse than the fine grid, with approximately double the fine grid's error percentages against the actual flight data. At the end of the experiment, the DES was deemed a more worthy turbulence model than any RANS model for aeronautical research, even with a simulation as complex as a complicated F-15E in a very high angle climb.

The DES has seen a lot more validation tests like flow cases over a rounded square and forebodies^[35], delta wing^[36], airfoils at high angles of attack^[37], a few other aircrafts^[38,39] and even supersonic flows^[40] and they have all attested to its superiority over the RANS and LES in functionality, accuracy, simplicity and robustness.

2.3.3 DES Issues

Yet the DES suffers from stark impediments from it being the perfect CFD solution to turbulence modeling. Menter *et al.* mentioned of a term called “grid-induced separation”^[41], which Spalart *et al.* later acknowledge as well as a problem with the DES^[30].

Since the DES is a RANS-LES hybrid system, it was expected that the method governing the determination of the location of flow separation near the boundary layer would be RANS. RANS does not require a high grid count within the boundary layer, that is to say that it allows cell dimension parallel to the surface of the wall to be relatively much larger than that perpendicular to the wall. And with Equation (13) and (14), so long as $d < C_{DES} \Delta_{DES}$ for the SA or $l_{SST} < C_{DES} \Delta_{DES}$ in the SST bases for DES, RANS is always the method choice for resolution of eddies.

Grid-induced separation, or GIS, happens at the point in between the RANS and LES approved grids whereby the length scale is approximately $C_{DES} \Delta_{DES}$. These grids were called “ambiguous” by both Menter and Spalart. They may also arise due to incorrect implementation of grid refinement (in the effort towards grid convergent study), or over-refinement in boundary layer regions where RANS should be used.

Ambiguous grids cause problems because grid spaces that are small inevitably resolve for lower eddy viscosity than required, encouraging larger eddies to grow and veering in the

direction of the LES. The phenomenon is called “modeled stress depletion” (MSD). Therefore under the DES scheme, when $\tilde{d}$ or $\tilde{l} \approx C_{DES}\Delta_{DES}$, it will always tend to select LES when in actual fact RANS should be still in effect due to the boundary-layer treatment. This results in a grid-induced early separation of flow.

2.4 Delayed Detached-Eddy Simulation (DDES)

MSD can be a very pronounced danger as it affects accuracy and precision in some cases, and inexperienced users may not even realize that their predictions have become erroneous. The problem is that no one exactly knows when the transition from RANS to LES takes place or how thick the boundary layer would evolve in any given simulation, much less for the inexperienced. The current method to solving this issue is to introduce a delaying feature to the length scale of the DES such that it prolongs separation point^[33]. This led to the derivation of the Delayed Detached-Eddy Simulation (DDES).

The best method so far was given by Menter *et al.*^[41] by using his two blending functions, F_1 and F_2 in his SST equations to determine the boundary layer regions, before approving or disapproving the use of LES, in that sense “protecting” the boundary layer. Although he has experimented and seen the success of his implementation in the DES which makes use of the SST as the subgrid scale model, or DES-SST, the concept is not just limited to just that turbulence model. In fact this was the correction required for DES on the whole,

regardless of the subgrid scale model, as long as the eddy viscosity term is the main working variable.

2.4.1 Derivation of DDES-SST

Menter's new blending function is given as

$$F_{DDES} = \max \left[\frac{l_{SST}}{C_{DES} \Delta_{DES}} (1 - F_{SST}), 1 \right] \quad (16)$$

And $F_{SST} = 0, F_1, F_2$. Recall Equations (3) and (5) that both F_1 and F_2 remain close to 1 within the boundary layer and becomes 0 outside the boundary layer. Thus when the DDES chooses F_1 or F_2 based on the call of SST, the separation location will be preserved as with proper SST solution even with “ambiguous” grids. And if there is no need for boundary layer shielding, the DDES-SST will naturally choose $F_{SST} = 0$, which returns the original DES-SST solution.

2.4.2 Derivation of DDES-SA

The SA equation is a single transport equation containing a comparison term, r , which is the ratio of the distance to the wall to its length scale. To add a delay to the DES-SA, Spalart made use of this ratio and turned it into

$$r_d \equiv \frac{\nu_t + \nu_m}{S\kappa^2 d^2} \quad (17)$$

The difference is that delayed ratio has an additional molecular viscosity ν_m added in the numerator such that at the wall $r_d \neq 0$. A delayed function is set up as

$$f_d = 1 - \tanh(8r_d)^3 \quad (18)$$

so that $f_d = \begin{cases} 1, & r_d \ll 0 \\ 0, & \text{otherwise} \end{cases}$.

Finally, the DDES-SA length scale is modified to be

$$\tilde{d} = d - f_d \max(0, d - C_{DES} \Delta_{DES}) \quad (19)$$

It is only when $f_d = 1$ that the LES condition is satisfied and allowed to be used. If $f_d = 0$, then $\tilde{d} = d$ and the SA solution is preferred.

In both of the methods above, the DDES is now corrected to predict more accurately the flow separation location, no matter how refined the grid is, by rejecting the tendency for LES to be implemented earlier than it should be.

2.5 Previous Work with C-130H

There has been a series of research by the United States Air Force Academy (USAFA) pertaining to the airflow around the C-130H, all part of the larger AIA project. Two of them are mentioned below.

Johnson *et al.* ran dye and particle image velocimetry (PIV) tests in the water tunnel on a 1:72 scale C-130H model with the cargo ramp lowered to attain the flow visualization aft of the aircraft^[38]. It was noted that there were strong regions of circulation just as flow left the tailgate and an apparent upsweep accelerating toward the bottom of the fuselage and tail was present thereafter.

Johnson *et al.* went on to perform a CFD analysis to visualize the spanwise vorticity of the flow around the tailgate region. In this segment, he solved the flow using the unsteady Navier-Stokes laminar setting in the flow solver *Cobalt* due to the low Reynolds number of 22700. Although this CFD experimentation had no direct application of turbulence in any way, it was an important study which helped in further expectations of turbulence in USAFA's near future.

The reason for doing so was because his setup of the PIV lasers and cameras could only view the flow from the side of the aircraft model. There was no way the setup could have enabled him to observe the tailgate flow squarely.

He found that two symmetrical spanwise vortices were formed at about the halfway point between the end of the cargo door and the tail due to the lower pressure regions at the mouth of the cargo by. These vortices traveled upwards toward the tail end, increasing their size while they were doing so. It was thus concluded that the traveling vortices were the cause of the upsweep.

With this knowledge, Johnson went back to his water tunnel model and built an extension to the cargo ramp. The ramp now had a length which was about double its original. With this, he discovered that there was less circulation at the end of the extended tailgate. The phenomenon of the upsweeping vortices was also greatly reduced such that the flow after the tailgate and near the underside of the empennage was minimally perturbed.

The *Cobalt* solver that Johnson *et al.* used is the commercial derivative of the military's cell-centered finite volume compressible solver *Cobalt₆₀*. It was developed by the Air Force Research Laboratory in the USA in the late 1990s^[43]. *Cobalt₆₀*'s algorithm was built using Godunov's first-order accurate, exact Riemann method. It also enabled parallel computing with the help of the ParMETIS domain decomposition library to divide a flow domain into several zones of similar sizes, which could then be passed to multiple processors via the Messaging Passing Interface (MPI).

In 2000, Cobalt Solutions, LLC took *Cobalt₆₀* and commercialized it under its new name, while at the same time upgrading the commercial version to a second-order accurate Riemann method and integrated the then new ParMETIS 3.0^[44]. *Cobalt* has been highly

successful with aerodynamic simulations and it has always been one of the USAF's and USAFA's top choices of CFD solvers. In the subsequent CFD experiments to be covered in this review, they were all performed on the *Cobalt* solver.

In 2003, Serrano in France and Leigh *et al.* in the USA took up where Johnson left off a year earlier. This joint effort purely focused on CFD analysis on the interaction between the C-130H and the extraction parachute in airdrop situations, particularly using the DES as its turbulence model^[45]. A rigid T-10 parachute was used as the extraction parachute the airdrop simulation. Investigations were set to 1000 ft MSL (mean sea level) and 140 KIAS (knots-indicated air speed) with a Reynolds number to length ratio of 1.488×10^6 1/ft.

The parachute had a towing line which was varied between 16 to 49 ft or 4.88 to 14.94 m from the end of the tailgate. It was shown that if the towing line was less than 30 ft, the canopy will be in the zone of the updraft. The DES was also able to show that the instantaneous turbulent vortices at the mouth of the cargo bay were not symmetrical, unlike when in the previous case by Johnson *et al.*^[38].

The next experiment was a comparison study between the wind tunnel hotwire results of the C-130H at École Nationale Supérieure d'Ingénieurs de Constructions Aéronautiques (ENSICA) and CFD at the United States Air Force Academy (USAFA). The ENSICA simulation was done by Bury while Claus *et al.* was in charge of the CFD aspect.^[46]

At ENSICA, a 1:48 scale model of a C-130H, with full wings and empennage, was subjected to a wind velocity of 40 m/s, which approximated to a flight Mach number of 0.12. Hot wire instruments were mounted in front of and behind the model aircraft in order to monitor the before and after flow aspects.

At the same time, at USAFA, an axially symmetric simplified CAD model of the aircraft was created. Claus *et al.* also used *Cobalt* as his solver with the DES turbulence scheme. He had a grid convergence study as well with two grids, the first having 4×10^6 unstructured cells and the second with 6.2×10^6 cells and remarked that there was an improvement in the results with the more refined grid.

He also noticed that there was a good resemblance between the CFD simulation and the wind tunnel PIV captures of upsweeping vortices propagating along the underside of the empennage into the wake. The DES also caught a downwash interaction of the wing tip vortices with the upsweep occurring slightly away from the fuselage. The only problem with the CFD study was that it was unable to depict clearly the empennage and upsweep vortex interaction possibly because the finer grid was still too coarse to resolve that. He indicated that a much more refined grid than 6.2 million cells was essential for subsequent segments of the AIA project. With this joint experimentation between USAFA and ENSICA, the AIA Phase I ended.

Morton *et al.* started Phase II of the AIA project, this time with only a 1:16 fuselage half model that has an opened and a closed tailgate configuration^[47]. They also ran a CFD with DES turbulence model on their CAD model and validated the results with wind tunnel PIV data from ENSICA, just as Claus *et al.* and Bury did in their experiment. With the main wings and the tip vortices induced by them, normally the aircraft would have to negate them by increasing its AOA to 2 to 8° so that the airflow angle seen from the cargo bay would be zero. Taking away the main wings removed the downwashing effects from the wing tip vortices, and thus the analyses could be done at an AOA 0°. In addition, Morton explained that the other reason for the removal the main wings, in this project and those prior to this, was that the 1:16 scale wing was too large for the wind tunnel width at ENSICA.

To conduct a more comprehensive grid convergence test than Claus did, Morton *et al.* had seven grids generated with increasing number of unstructured tetrahedral cells, from 3.05×10^6 to 11.27×10^6 . They chose the Spalart-Allmaras with rotation/streamline curvature correction model (SARC) over the SST turbulence scheme because it has been found that the solutions derived with the SST did not portray the curvature effects of convex walls on turbulence as the SARC does^[6]. In addition although the SST was found to be 1% more accurate than the SARC, the 1% improvement in solution is small compared to the 14% increase in computation time required. At the same time, they did a comparison of the SARC with the DES-SARC to from angles of attack -5 to 10°.

They showed both the RANS and the DES methods produced very similar lift and drag coefficients. However, that was the end of the similarities between them. The DES produced visuals of ± 750 rotations/s which depict far more turbulent structures streaming from the aircraft body and into the wake than the SARC. In fact the SARC produces has too much artificial viscosity built into its algorithm such that it dissipates a lot of vortex energy. Hence the DES only affected the solution in terms where instantaneous turbulence were needed to be understood.

Moreover the DES was able to predict the magnitude of and location at which the upsweeping vortices form along underside of the aircraft very well, according to the PIV visualizations, although the shapes of those vortices were slightly incorrect. On the other hand, as the upsweep break up to form smaller trailing vortices with surrounding detached vortices in the wake, these were much better modeled, magnitudes, shapes and sizes which agreed with the wind tunnel tests.

A continuing research by Bury *et al.* in 2008 showed more information of the airflow at the near wake of the simplified C-130 fuselage^[48]. In the closed hatch scenario, they tracked the trajectory of the vortices formed on the underside of the fuselage and into the wake at 0° AOA. The upsweep vortices followed very closely to the empennage geometry before emerging horizontally into the wake. Induced vortices, created by the wall-upsweep vortex interactions, were birthed just under the empennage's horizontal wings and the travel downwards in the wake, decaying rapidly as they leave the empennage zone. They also

calculated that the upsweeping vortex intensity was 2.25 times that of the induced vortex's due to a 9.5% increase in turbulence intensity than with the latter.

Tracking the vortices' movement with the opened hatch configuration showed that upsweep was formed much later, right after the ramp, and it had a helical trajectory. The core of this vortex also revealed that it had higher turbulence readings too. On top of that, two smaller and weaker symmetrical vortices from under the ramp were seen to add to the turbulence in the near wake, weaker due to the fact that the velocity has been reduced in the region.

Finally, in 2009, Bergeron *et al.* worked on the same 1:16 computational model of the Hercules fuselage, this time replacing the DES-SARC with DDES-SARC, since DES can incur problematic solutions in thick boundary layers^[49]. They used the *Cobalt v4.2* solver which by that time had converted all the DES models into DDES. Two grids were used to perform a grid sensitivity study: one with 8.8×10^6 and another with 10.5×10^6 number of cells. They also had three time steps sizes, 0.001 s, 0.0002 s and 0.0001 s, to balance the Courant number.

The remarkable feature they found was that when comparing the DES, DDES and PIV data of vortices evolved aft of the aircraft, the DDES portrayed the vortex shapes and orientation which were almost exact to the PIV. The DDES also gave a more detailed view of the turbulent structures forming from the underside of the opened cargo ramp as opposed to the DES before in Morton *et al.*'s^[47] findings.

However, with that said, predictions of the starting location of flow separation and the vortex strength still displayed some errors. DES previously could not determine where flow separation occurred, although it showed where the vortices were born, and now DDES still faces the same situation. Vortex strength was over-predicted as well when the cargo hatch was down. Bergeron commented that this could be because of two reasons. The first was that the longitudinal pressure gradients were high because of the lack of wing tip vortex interactions with the empennage. The second was due to the turbulence model, SARC. In regions where the grid has already been resolved to capture the major flow features, SARC has a tendency to overcompensate, causing the solution to deviate from its true value.

As a result of the review of work done on the C-130H so far, there is still room for improvement for a more accurate turbulence model algorithm than just the DES and DDES. Yet, at the moment, the DDES is considered the most robust and widely solver-integrated RANS-LES hybrid in the market, and will probably be so in the near future. The research has also been concentrated on the SARC implementation of the DDES while very little has been done with the SST, which is a viable choice in aerodynamic considerations as well. Therefore it might be beneficial to run a more holistic simulation just with the SST to give a clearer picture of how it differs from the SARC, especially since SARC can potentially paint an erroneous picture with high grid resolutions.

3. NUMERICAL SET-UP AND METHODS

3.1 Phase I – 1:16 C-130H Fuselage Model

3.1.1 Flow Domain and Grid

The first phase in this numerical analysis project of the C-130H was a comparison study of the DDES-SARC's and DDES-SST's predictions of the airflow over the fuselage. This was essentially a reproduction of the CFD experiments that both Morton *et al.*^[47] and Bergeron *et al.*^[49] did. It also served as a validation of the *Cobalt v5.0* solver and its contrast with *Cobalt v4.2*.

Cobalt Solutions, LLC upgraded the *Cobalt v4.2* to *Cobalt v5.0* by changing all the DES schemes (DES-SA, DES-SARC and DES-SST) to DDES schemes in light of its pronounced improvements^[44]. Although Bergeron implemented the DDES-SARC in the fuselage study using the *v4.2* solver, the algorithm then was still a beta release version before the actual release in *v5.0*. Thus the new algorithm might have had some minor tweaks to ensure it was accurate. Among other modifications to the *v5.0* were the enhanced actuator disk boundary condition and the reduced overlap requirements in overset grids.

Cobalt has its limitations though. It is primarily an aerodynamic finite volume solver for subsonic, supersonic and even some hypersonic research. As the software code is proprietary and closed, end-users may find it difficult to manipulate flow and system

equations, perhaps add new parameters or even change the type of fluid used. To add to that, grids that have multiple block domain were not supported. *Cobalt* allows multiple domains only if these domains were superimposed as overlap grids on the main domain grid.

For this phase, USAFA has provided a similar geometry and domain mesh as the one which had already been used earlier on. The fuselage nose-to-tail length is 170 cm, making it about a 1:16 scale to the actual length. The flow domain measures $4000 \times 2000 \times 1000$ cm and the model fuselage is located at about 10 times the fuselage length from the inlet.

Figure 3: Dimensions of the Phase I flow domain in which the C-130H fuselage sits in.

Figure 4: Phase I's C-130H fuselage model.

The domain grid had 12.8×10^6 tetrahedral cells, a 2.3×10^6 increase in cells than that used at USAFA. Unstructured grids are easier to generate than structured meshes and they have a higher degree of fidelity to the geometry too because the cells have little difficulty with fitting into tight areas. In the *Cobalt* check, the grid had a grid quality of 94.6/100, meaning that there was virtually almost no cell in the domain which was skewed or anisotropic. A good unstructured mesh is one where the cells are generally isotropic or in other words, every cell's dimensions were about equal. Isotropy allows for more accuracy in calculation of flux terms in and out of the control volume, thus maintains a proper solution even where there are adverse changes in the flow properties.

Below are two figures showing the grid refinement from freestream towards the boundary layer and the wake region. As USAFA had no expectation of how the turbulent structures might develop *a priori*, the grid was set to have a cell growth rate as it extends beyond the

thin shear layers near the wall. The growth rate however was not indicated when USAFA presented the grid for this project.

Figure 5: Mesh of the flow domain on the symmetry plane.

Figure 6: A close-up view of the empennage end mesh.

Figures 7 and 8 show the surface mesh of the C-130H's fuselage. Note that the entire empennage region saw a greater grid refinement than the rest of the body did. This was to

ensure that the level of detail captured at the areas aft of the aircraft would be of high accuracy, since this is the main point of interest in the experiment.

Figure 7: Plan view of the C-130H's head's surface mesh.

Figure 8: Plan view of the C-130H's empennage.

3.1.2 Numerical Settings

The conditions were essentially the same as the wind tunnel experiment at ENSICA: $\gamma = 1.4$, $T = 298$ K and the atmospheric pressure was $p = 101325$ Pa. It was assumed that the wind tunnel was in room environment which had no major difference from that at 0 MSL. By setting the temperature and pressure, *Cobalt* automatically decides two other major flow properties: speed of sound a and the density of the atmosphere ρ . They are given in accordance to the 1976 Standard Atmospheric Table, together with the assumption that air is a perfect gas, then the corresponding speed of sound was $a = 346.07$ m/s and atmospheric density was $\rho = 1.1844$ Pa. The body of the fuselage was taken as a smooth, adiabatic and no-slip wall. No trips were implemented on the body surface too.

The airflow was staged at $M = 0.1155$ and flowing in the negative x-direction, producing a flight Reynolds number of $Re = 4.382 \times 10^6$. For *Cobalt*'s computational purpose, it calculated a per length Reynolds number of $Re/L = 2.578 \times 10^6 \text{ m}^{-1}$. Simulations in this phase were done with the *Cobalt* solver software for mainly two reasons: firstly, it provided a good platform to test *Cobalt v5.0*'s capabilities and, secondly, the grid provided was of a file extension native to only *Cobalt*, hence only compatible with *Cobalt*.

1500 iterations first were run with 1 Newton sub-iteration per iteration and at the default timestep set by *Cobalt*. This was done so that the simulation might stabilize before starting the solution process proper. The Courant-Friedrichs-Levy (CFL) number was initially set at

10 and was ramped up to 1×10^6 within the first 50 iterations. It was then maintained at that number for the rest of the 1500 iterations.

The actual solution, as it was to be time-dependent, the number of sub-iterations was increased to 3 to improve accuracy while the timestep was reduced to the designated constant value of 0.001 s. This was to be maintained until the solution converged. Temporal and spatial accuracies have also been set to second-order accuracy while advection and diffusion coefficients have been put to 0.01 and 0.05 respectively. The advection and diffusion coefficients affect the stability of the simulation as well as the time taken for convergence.

The simulation was run on eight 3 GB core nodes on the IBM high performance computer (HPC) at the Nanyang Technological University's HPC Centre. Each of these cores are a bundle of 8 processors, therefore the total number of processors working on a simulation at any given time is 64 with a total of 192 GB worth of memory. To complete 1000 iterations of computational solutions on this massive domain required 7.8 hours.

3.2 Phase II – Full-sized Half-symmetric Aircraft Model

The second half of this project looked into the effects of tailgate on the mouth of an opened cargo bay. The main wings have also been added to C-130H model to observe their effect on recirculation zones evolved aft of the aircraft.

3.2.1 Flow Domain and Grid

This new model was no longer provided by USAFA but was generated by Centro Internacional de Métodos Numéricos en Ingeniería (CIMNE[®]), which is an international company engaged in numerical methods in engineering research. The grid was created by GiD, primarily a finite element pre and postprocessor that is compatible with several finite element solvers like NASTRAN, but it too can create meshes that are suitable for finite volume solvers like ANSYS FLUENT^[50].

The GiD suite was developed by CIMNE[®] as its unique means to provide a fully packaged software to render greater control over the entire computational solution process. The pride of GiD's preprocessing capabilities is that it has the ability to generate large meshes of good quality quickly. It boasts of three algorithms which it utilizes in order to build volumetric tetrahedral meshes: advancing front technique, Delaunay technique and the Isosurface algorithm. GiD is also proficient in producing grids with multiple domains for better cell and element manipulation control in regions which require that.

To reduce the number of grid points and hence computational resources for the Phase II project, only a half model of the C-130H was generated, with the symmetry along its longitudinal axis. The model is a 1:1 scale of the actual aircraft and it had more details on this than USAFA's. The main wings were added and the nose was properly crafted, although Bury had commented that the nose played no effect on the vortex build up at the

empennage^[48]. The turboprops have been mounted on the wings, however the propellers have been omitted, therefore only the nacelle was present.

Two configurations of the aircraft have been put forth as well: a closed and an opened cargo hatch. The closed hatch would be able to give the contrast of the main wings' effects on the empennage. Then with that information, it can show the difference between an opened and a closed tailgate.

A representation of the flow domain with the aircraft model is provided in Figures 9 and 10 below. The flow domain is $210 \times 90 \times 30$ m and the nose of the aircraft is situated 75 m from the inlet along the length of the domain. 105.2 m of the domain still exists behind the model so as to allow the wake to be fully formed.

The inlet was set at the front face of the domain box facing the model's nose, that is in the negative y-direction, the outlet at the back and the rest of the sides of the domain were treated as the farfield boundaries. As with Phase I, the body of the aircraft was taken to be an adiabatic wall that was smooth.

Figure 9: Phase II's C-130H in its computational domain.

Figure 10: Dimensions of the computational domain from the back view.

The experiment was performed on two grids, one with two computational domains and the next with three. The former grid had 1.28×10^6 number of cells and the farfield was filled with a coarse mesh. Nearer the aircraft, an area with a higher concentration of cells was drawn around the aircraft, thus forming two coherent domains within the flow field approaching the aircraft.

Figure 11: Double-domain Phase II mesh.

The latter was a triple domain grid, with a total of 3.36×10^6 cells, which had much higher refinement of the mesh near the cargo bay. This enabled a semi grid convergence study pertaining to the grid density required to understand the vortices evolving in the near-wake region.

Figure 12: Third domain in Phase II's triple-domain mesh.

3.2.2 Numerical Settings

ANSYS' FLUENT 13.0 was selected as the finite volume solver for this phase because of its compatibility with CIMNE's pre-processed grids. Due to the multiple domains in the grid, *Cobalt's* code could not accurately interpret the continuity at the inter-domain interfaces.

Although it would have been best if *Cobalt* could be utilized as the solver in the second phase to maintain uniformity in solution methods, FLUENT still served as an adequate substitute. FLUENT is among one of the most popular commercial solvers because of its simplicity in usage, its diverse solver codes which enable it for many areas of CFD analyses and that it already has its own post-processor built in for direct interpretation of results. On a note more related, ANSYS released FLUENT 12.0 in 2009 with the delayed option included for all its DES turbulence models and this feature has also been implemented in the current thirteenth version of the solver^[51]. The choice of DDES in Phase II's simulation was a reaction to the benefits of this turbulence model, which has already been pored extensively over in Chapter 2 of this report, over the DES, LES and RANS models.

Dimensional analysis was not performed in any of the phases. The fundamental reasons were that dimensional analysis violated the limits of airflow physics and solvers' capabilities. Firstly, in Phase I, for a proper comparison between computational model and the actual aircraft, the airspeed experienced by the model has to be 16 times faster. As *Cobalt* is an aerodynamic solver that only contains a single fluid medium (air), this equated to a hypersonic speed of 1152 m/s for the typical actual speed of 140 knots or 72 m/s. There is no physical way to impose such speeds unless the density of the fluid medium is changed.

In a similar fashion, Phases I and II cannot be directly related as the 40 m/s airspeed in Phase I would translate to 2.5 m/s in Phase II, which does not make any physical sense. Hence the simulation will be conducted speeds of 40 and 72 m/s at sea level density and

then at a situation more similar to actual airdrop conditions where the altitude is 1500 ft. The 72 m/s is the typical airdrop speed for the standard airdrop method with the C-130. At the sea level setting, $T = 298$ K, $\rho = 1.225$ kg/m³ and $p_G = 0$ Pa. At the standard airdrop altitude, $T = 285.18$ K, $\rho = 1.172$ kg/m³ and $p_G = -5353.21$ Pa.

4. RESULTS AND DISCUSSION

4.1 Phase I Results

As *Cobalt* is purely a CFD solver, *Tecplot 360* was the software of choice for post-processing the results. *Tecplot* may be considered to be a versatile universal visualization tool for many numerical simulation solvers as well as a reader for any charts and data sheets produced by the solvers at the end of their tests. The software is easy to learn and simple to use, making it an ideal choice for both amateur and professional numerical analysts.

Streamlines near the symmetry of the fuselage revealed that the Coanda effect drew the airflow inwards from the sides and the underside along the curvatures of the fuselage into the aft region. This produced an upwards flow, as depicted in Figure 13 which was observed in Johnson *et al.*'s^[38] experimentation.

Figure 13: Streamlines of airflow around Phase I's C-130H fuselage.

The time-averaged lift (C_L) and drag (C_D) values over the range of AOA tested showed that both DDES-SST and DDES-SARC turbulence models did indeed produce very similar results. The only exception was an outlier at 5° where the DDES-SARC measured a substantial 0.003 difference in C_D value from the DDES-SST. Comparing them with the wind tunnel test results given in the report by Morton *et al.*^[47], the percentage inaccuracy in C_D of the DDES-SARC is 30% while the DDES-SST is only 6%. Table 1(a) and (b) give the C_L and C_D comparison data between both numerical models.

Table 1(a): Comparison of C_L between DDES-SARC and DDES-SST.

α	DDES-SARC	DDES-SST
-5	-0.189	-0.196
0	-0.0731	-0.0734
5	0.0498	0.0489

Table 1(b): Comparison of C_D between DDES-SARC and DDES-SST

α	DDES-SARC	DDES-SST
-5	0.0290	0.0305
0	0.0132	0.0135
5	0.00817	0.0113

Figure 14: Graphs of lift and drag coefficients with varying AOA from -5° to 5° .

The stark difference in results at this AOA could be the way the SARC accounts for the rotation compensation and in so doing, produces a lower C_D value than expected. This could be more clearly seen by investigating the vorticity evolving in the near wake in the segment below.

The vorticity equation is defined as half the curl of velocity, or

$$\omega = \frac{1}{2}(\nabla \times u) = \frac{1}{2} \left[\left(\frac{\partial u_z}{\partial y} - \frac{\partial u_y}{\partial z} \right) \hat{i} + \left(\frac{\partial u_x}{\partial z} - \frac{\partial u_z}{\partial x} \right) \hat{j} + \left(\frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y} \right) \hat{k} \right]. \quad (20)$$

In the earlier report^[52], the DDES-SARC x-vorticity magnitude was measured as ± 10 rotations/s, about 100 times lower than that from Bergeron *et al.*'s^[49] findings. That was a

mistake however because the model was built and measured in centimeters at USAFA. The x-vorticity result was justified because Bergeron *et al.* referenced his x-vorticity to meters. Hence in order to make qualitative comparisons and analyses with their data, all readings in this report were multiplied by 100.

Figure 15 shows the corrected vorticity plots of the C-130H of vorticity magnitude of 750 rotations/s at zero AOA. It was found to have the same turbulent effects emerging from the tail area – upsweep, detached and tailwing tip vortices – except that the vortex pockets feeding the upsweep vortices were not as pronounced as Bergeron *et al.*'s. In the same AOA = 0° case, the DDES-SST revealed that each upsweep vortex was accompanied by more detached vortices.

Figure 15: Iso-surface plot of x-vorticity with DDES-SARC of magnitude of 750 revolutions/s at 0° AOA. Colours of absolute pressure measured in Pa.

The major differences between both the DDES-SARC and DDES-SST numerical models surfaced in the manner of the formation and propagation of the turbulent vortices from the empennage surface into the near wake. As seen from the DDES-SARC result of the x-vorticity iso-surface, each upsweep vortex on either side of the tail was accompanied by a single detached vortex, whereas the DDES-SST displayed at least three other detached vortices following an upsweep vortex, as shown in Figure 16. Besides that, the vortex pockets on the underside of the aircraft were also larger and more defined.

Figure 16: Iso-surface plot of x -vorticity with DDES-SST of magnitude of 750 revolutions/s at 0° AOA. Colours of absolute pressure measured in Pa.

Similar to the effects seen at 0° , at -5° the DDES-SARC gave a decent account of the vortices evolved, even showing how the vortex pockets detached themselves from the surface at early onset to form another pair of detached vortices. Unfortunately, the same numerical model could not resolve any form of turbulence in the wake at 5° . The wake in this instance was too diffused such that there was little clarity as to what the fuselage was facing.

These results were in no way comparable to that given by the DDES-SST simulations at the two mentioned AOA. The level of detail was by any length greater in both cases and in the -5° scenario (Figure 17), an evidently turbulent wake describing the unsteadiness and breakdown of vortical structures was portrayed.

Such observations were in line with Zhong's RANS comparison of the SST and SARC in his study on vortices borne from a wing-fuselage interaction at various AOAs^[53]. He pointed out that the SA equation tended to induce large dissipations near the aircraft surface, therefore reducing the clarity of vortex cores and was ultimately inadequate in resolving the vortices accurately.

Since the SARC was built upon the SA equation by adding the rotational correction function, there was no change in the way the dissipation, governed by ν_t , was modeled. Although the SARC did improve how fluid flows reacted with the curved surfaces and could present a closer prediction to the actual wind tunnel experimentations, it still did not address the issue with the dissipation rate experienced in the SA. Zhong realized that on the whole, Menter's SST was still the better turbulence model to use because of its close semblance of evolved vortices and lift and drag values to his wind tunnel results.

Figure 17: Iso-surface plot comparisons between DDES-SARC and DDES-SST at (a) 5° and (b) -5° .

There is also another reason to the difference between both models. Menter in his SST formulation had already accounted for the greater production of turbulent shear stress over its dissipation in the boundary layer. He ensured that the eddy-viscosity would observe Bradshaw's assumption of a proportional shear stress to turbulent kinetic energy. Elsewhere in the free shear flow, the eddy-viscosity would be allowed to return to its

original value of $\nu_t = k/\omega$. The effect was that in regions of adverse pressure gradients, the SST could manage the eddy-viscosity properly in order to produce the desired predictions.

The SARC however did not have such an implementation in its single transport equation because the eddy-viscosity was not modeled to suffer such adverse pressure situations. Even after a pressure drop from a shock, Spalart and Allmaras found that the vorticity magnitudes behind the shock were on the whole too small to produce much eddy-viscosity, thus they likened it to a normal shear flow^[5]. The consequence is that when scrutinizing the empennage region where pressure gradients are high, the SA, and in that matter the SARC, inaccurately models the turbulent flows. Just as Bergeron *et al.* said, a grid that is sufficiently resolved to capture the turbulence of curved surfaces, the SARC equation has a risk of over-compensating on the rotational eddy-viscosity to produce undesirable results.

Thus, just by comparing both turbulent numerical models, it is evident that the SST, not the SARC, proved to be the more appropriate model to simulate the effects of an opened cargo hatch. It has already been shown that the SST with DDES implementation was able to reveal a lot more detail of the vortices evolved in the wake, and this promises to provide a more comprehensive study of what the parachutes and goods feel during any airdrop procedure via the tailgate.

4.2 Phase II Results

In order to find out whether the double-domains or the triple-domains mesh was more useful for simulations, a comparison experiment at 0° with an airspeed of 40 m/s was performed on both of them, with their aircraft's cargo hatch lowered. Since the DDES-SST showed more realistic results in Phase I, the same turbulence model was applied back here in Phase II.

The double-domain mesh was unable to resolve any vortex core, or for that matter turbulent eddying, at the exposed cavity region of mouth of the cargo bay. On the other hand, the triple-domain grid, due to its higher density of cells focused at the area of concern, presented excellent details of the size of the vortices and the extent of the recirculation zone. This contrast may be seen in Figure 18.

Figure 18: Comparison of 100 rotations/s iso-surfaces at 40 m/s with the double- and triple-domain meshes.

The drawback of choosing the triple-domain mesh so as to produce the level of detail required was that it took nearly 24 hours on 32 processors to converge while the double-domain required merely half the time. Unfortunately, there was no middle ground to strike between the computational and detail requirements, the triple-domain mesh was the only choice available.

The vorticity magnitude of 100 rotations/s here versus the 750 rotations/s in Phase I's results was an effect of the change in dimensions of the model and the speed at which it

was travelling at. Bearing in mind that as no dimensional analysis was performed, Phase I and Phase II had almost no direct relationship in terms of these numbers.

With the main horizontal wings attached to the C-130H fuselage, it was expected that the aircraft would experience a dramatic change in lift and drag. In the first phase, it was observed that the drag was lowest at around 2° . However with the main wings in place, the drag peaked at an AOA of slightly less than 0° , as seen in Figure 19, while the trough still required more tests at high AOAs to be determined. The lift saw a stark increase in gradient too, but it was impossible to note the magnitude of that gradient increase since both cases were entirely different.

Figure 19: Lift and drag coefficient graphs of the C-130H with main wings at 40 m/s.

Morton *et al.* had already put forth that with the main wings, the downwash effect would affect the tail region in such a way that in order for an efficient airdrop, the angle of attack

of the aircraft must be within 2° to 8° ^[47]. Figure 20 below show the velocity magnitude contours towards the back of the fuselage in Phase I at 0° . It is noticeable that there was very little disturbance in airflow as it moved towards the empennage area as seen from the smoothness in the contour lines.

Figure 20: Contour lines of velocity magnitude on the C-130H fuselage in Phase I.

In Phase II where the main wings are present, velocity vector plots in Figure 21 present the downward velocity 2 m behind the main wings. The freestream velocity for this and subsequent cases was set to 40 m/s. The area of congested arrows near the fuselage body is just the result of the domain of refined mesh around the cargo bay mouth. The Coanda effect slightly redirected the flow upwards as it approached the curved wall of the fuselage bottom but not enough to negate the downwash.

Figure 21: Velocity vector plot of the downwash aft under the empennage in m/s. $AOA = 0^\circ$.

Vector length = 0.4 grid units/unit of vector magnitude.

The contour shapes from the downwash effect (Figure 22) were more uneven than without the main wings. There were also smaller vortices induced and propagating upstream parallel to the wall. The good thing about the main wings' downwash was that it caused a slowing down of the transition from 0 m/s to freestream velocity (seen by the spaced out contour lines) as opposed to the sudden jump in Phase I near the fuselage body.

Figure 22: Contour lines by velocity magnitude on the C-130H in Phase II. AOA = 0°.

Figure 23 shows the velocity vector plot of an AOA of 5° again at 2 m behind the main wings. The increased AOA negated the downward forces felt aft of the aircraft and in so doing, removed the small parallel vortices propagating under the fuselage surface as seen in Figure 24.

Figure 23: Velocity vector plot of the downwash aft under the empennage. AOA = 5° .

Vector length = 0.4 grid units/unit of vector magnitude.

Figure 24: Contour lines by velocity magnitude on the C-130H in Phase II. AOA = 5°.

When the tailgate was lowered, low pressure areas created in the portion of space confined by the underside of the empennage and the cargo ramp caused high pressure airflow at the side and under the ramp to rush into the space. This formed a large symmetrical counter-rotating vortex sheets on either side of the ramp. The effects of the raised angle of attack are seen more vividly (Figure 25(b)) with these vortices as illustration.

At an $\text{AOA} = 5^\circ$, the vorticity iso-surface plot of 120 rotations/s found that the vortex core was slightly tighter and closer to the tailboard and produced smaller turbulent swirls at the edge of the tailboard than in the case where $\text{AOA} = 0^\circ$. Suction pressures ($p_G < 0$) in the increased AOA case were also closer the mouth of the cargo bay while the region just beyond the tailboard's edge experienced pressures of positive magnitudes. This could be important for the parachute inflation during the extraction process.

Figure 25: Vorticity iso-surface plots at 120 rotations/s at the mouth of the cargo bay with AOAs of (a) 0° and (b) 5° . Contour colours by gauge pressure measured in Pa.

Having a contribution of downwash from the main wings might not necessarily be a problem for airdrops. In fact, the downwash caused very little alteration to the velocity and forces at the cargo bay that it could be negligible due to the upsweeping forces which are prevalent under the fuselage. From the view of the cargo bay, the negation of the

downwash just enabled parachutes and goods to be released at an angle that was more level to the ground, especially at low altitude airdrops. In fact, the angle of which the cargo ramp was lowered might change the trajectory of airdropped goods more than the downwash did, as in Figure 26, however the scope of the project does not cover this aspect of the protocol.

Figure 26: Streamlines on the underside of the aircraft demonstrating the change in airflow direction.

Opening the cargo hatch caused some drastic changes on the lift and drag coefficients. This is depicted in the charts plotted in Figure 27. At 40 m/s, without lowering the hatch, the drag for the half C-130H model peaked at -1° with a value of approximately 0.115 and the lift curve gradient was about 0.32. The drag curve continued to have its maximum at -1° when the cargo ramp was lowered and with a dip in drag coefficient to 0.05. Between the AOA of -6° to 3° , the C-130H could possibly enjoy a benefit of less drag by opening up

cargo bay than if it were closed. Unfortunately the range at which the aircraft felt higher lift when it lowered its tailboard was when the AOA was less than -2° , at which there is zero lift. The overall lift gradient for opened cargo hatch was 0.099.

Figure 27: Lift and drag coefficient graphs comparing an opened and closed cargo bay.

The more important scenario in this project was the case when the C-130H aircraft was at the typical airdrop speed of 140 knots and at the standard airdrop altitude of 1500 ft. Changing both these parameters of flight altered the physics of turbulence experienced under the empennage.

For the 0° AOA situation, at a vorticity magnitude of 120 rotations/s, the core of the vortex was much larger and voluminous while the pressure difference range was at least 4 times

more than when the aircraft was simulated at sea level and at 40 m/s flight speed. The same could be said about the iso-surface plot of the 5° AOA case. In this instance however, by increasing the AOA the vortices expanded especially near the end of the tailgate. Swirls of airflow were also more prominent as it departed from the tailgate into the near wake under the empennage. Figures 28(a) and (b) illustrate the observations. This could be explained by looking at the velocity plots on the plane at the tailboard's edge.

Figure 28: Iso-surfaces of vorticity magnitude 120 rotations/s at (a) $AOA = 0^\circ$ and (b) $AOA = 5^\circ$.

In the $AOA = 0^\circ$ case, present at the edge of the tailboard was a vortex core that was made up of least five counter-rotating vortices tightly bound together. The overall effect was that as a group (after attempting to cancel each other out in magnitude) they still possessed a general weak clockwise rotational pattern.

(a)

(b)

Figure 29: (a) Velocity contours and (b) vectors of the vortices evolved at the tailgate's edge. $AOA = 0^\circ$. Vector length = 0.4 grid units/unit of vector magnitude.

With a 5° angle, the vortex core was now no longer a conglomerate of smaller vortices. A single vortex, unhindered, might be allowed to grow until it encountered an obstruction in its rotation. In both cases though, it was undesirable to release the parachutes so close to the edge of the tailgate because the over-arching large vortex cores present which might spin an inflating parachute and twine the suspension lines together such that the parachute never fully inflates. Furthermore, the risk of the upsweep was very present at the edge and it will be easy for any parachute to be caught in it.

(a)

(b)

Figure 30: (a) Velocity contours and (b) vectors of the vortices evolved at the tailgate's edge. $AOA = 5^\circ$. Vector length = 0.4 grid units/unit of vector magnitude.

Johnson *et al.*^[38] mentioned in the experimentation that if the tailgate were to extend to double its length, the extraction parachute would not be caught in the upsweep. The principle behind this method was to release the parachute downstream far enough such that it would not feel the effects of the upsweeping forces. There was thus a window of space in which if a parachute is caught within might cause a failed extraction process. Therefore it is highly probable that even without the extensible tailgate, goods may still be deployed if the parachutes are launched beyond this affected region.

The attempt to find this safe point brought the project back to observing the velocity vectors and streamlines that propagate away from the cargo bay. Releasing streamlines laterally outwards from the axi-symmetric plane show how the fluid particles travelled from the nose of the C-130H, past the mouth of opened cargo bay and into the wake of the aircraft. The point at which parachutes would no longer experience the upsweep was estimated as the point where the streamlines started to realign with the freestream airflow again.

As was mentioned, the lowered ramp had an effect on the airflow as it followed the curvature of the ramp closely. As it passed the ramp into the region beyond the ramp's edge, by the Coanda effect again, the flow was also initially redirected upwards, as Figure 26 has already pointed out. This was not necessarily in the direct influence of the upsweep felt under the empennage, nonetheless, this upwards drive from the tailboard might push parachutes into the upsweep. Therefore, as a precautionary measure, the safe zone will have to be taken downstream from the said phenomenon.

With the $AOA = 0^\circ$ case, this point was at a distance 3.85 m from the edge of the tailgate edge. With reference to the empennage, this would also correspond to about 3.43 m from the end of the tail. It was also approximately 1 m in vertical displacement below the tailboard edge. Figure 31 shows this position.

Figure 31: Measurement of the distance of zero-upsweep from the tailboard's edge.

$$AOA = 0^\circ.$$

So far only the longitudinal distance has been checked. The height of the window also had to be ascertained. A velocity vector plot at that 3.85 m mark from the tailboard revealed that there was a large rotating zone present. From the spot where vectors started turning downwards to the point previously mentioned, that distance was found to be 2.84 m. In effect a parachute should not experience any upwards force beyond this reference “zero-upsweep” or ZU zone.

Figure 32: Measurement of the height of zero-upsweep window. $AOA = 0^\circ$.

Vector length = 0.4 grid units/unit of vector magnitude.

However, the parachute diameter would also affect its likelihood of being caught in the updraft. For example, a G-12 circular parachute has a D_0 of 64 feet or 19.5 m and a $D'_p = 0.67D_0$ or $D_p = 13.07$ m, and that would mean an inflated projected radius of about 6.54 m. Assuming that the centroid of the parachute were to be right at ZU point, the radius would not still fit the height of the zone. In such a case, the airdrop specialists would have to release the suspension lines further to allow safe ejection of goods. A C-9 parachute with

an inflated projected radius of 2.84 m would be the perfect parachute to be launched at the threshold of the ZU zone.

At an AOA of 5° that ZU zone was 7.29 m from the tailboard edge, which was twice the distance than the 0° scenario. The height of this window was, on the other hand, only slightly larger, measuring about 3.41 m. Likewise, any parachute released and inflated downstream of this point would likely be out of the upsweep influence.

Figure 33: Measurement of the distance of zero-upswEEP from the tailboard's edge.

$$AOA = 5^\circ.$$

Figure 34: Measurement of the height of zero-upsweep window. $AOA = 5^\circ$.

Vector length = 0.4 grid units/unit of vector magnitude.

5 CONCLUSION AND FUTURE WORKS

The C-130H, “airdrop work horse”, has had countless researches to improve the aerodynamics, the models and variants and also the airdrop systems from the aircraft. In fact, as seen in Johnson *et al.*'s^[38] work, this paper was not the only one which proposed a method to protect the parachute from the upsweep which plagues the extraction methods currently devised. However, this paper inspected the three-dimensional airflow around the opened cargo area and just as it left the tailgate in order to better understand and provide more qualitative suggestions for better airdrop procedures.

CFD has been a cost-effective and efficient means of testing the C-130H in multiple scenarios of various AOA, at different altitudes and with two flight velocities. The concern for the use of CFD was then to select an appropriate unsteady turbulence model which was also three-dimensional to best represent the actual events that might occur during a typical flight.

In the process of deciding on that turbulence model, it was found that the DDES-SST was superior to the DDES-SARC in a number of aspects. The DDES-SST was able to firstly predict lift and drag coefficients that were closer to actual test results whereas the DDES-SARC, although similar on most angles, at the 5° AOA had remarkably deviating results. The iso-surface figures of vorticity magnitude next confirmed that the DDES implementation of the SARC clearly misinterpreted the aerodynamics due to the over-

compensation in the turbulent model's code and algorithms. The former turbulence model fortunately portrayed a lot more fidelity to the details of vorticity forming and unsteadiness.

Although Morton *et al.*^[36] has briefly mentioned that there was a slight improvement of the DDES-SST over the DDES-SARC while performing his simulation, their report did not provide the actual comparison data from both DDES models. This project is currently the first in presenting the differences between these two versions and also in implementing the DDES-SST on the C-130H, which no military scientist has ventured into to date.

By observing the vortices formed at the mouth of the cargo bay and how these vortices travelled downstream, a zone could be cordoned off and labeled as “unsafe” for parachute deployment. In this zone, airdropped goods may experience the undesirable consequence of the upsweep flow occurring directly under the empennage surface. Conversely, parachutes launched downstream of this point may be considered safe for continuing the extraction process provided the size of the parachutes were taken into account.

There were several limitations encountered in this project. The first was the lack of an opened cargo hatch model in *Cobalt*. Without such a model, this project simply could not have continued since the objectives were to attain an understanding of the airflow at the region of the opened cargo bay and then determine an acceptable extent for safe airdrop purposes. As a result, CIMNE[®] was approached to construct a lowered cargo ramp model.

This first limitation inevitably led to the second, which was about the issue of congruency with the two solvers. *Cobalt* only accepts grids and computational models generated by a few pre-processors like VGRIDns, Gridgen and Pointwise, thus the model and flow domain mesh created by GiD could not be recognized. Although it was possible to convert the mesh from the FLUENT format to a *Cobalt*-accepted nature, the multi-domain functionality of the mesh hindered *Cobalt* from producing any comprehensible solution. The reverse for FLUENT is true as well because FLUENT does not have the ability to read *Cobalt*-designed meshes.

Cobalt is a solver solely built for aerodynamic purposes and it excels in that area of interest. FLUENT on the other hand, though it has the capability for aerodynamic CFD research, is a general CFD solver. There were aspects in FLUENT which did not work as well as *Cobalt*, for example the interpretation of vorticity iso-surfaces. As remarked in Chapter 3 that it would have been best to continue using *Cobalt*, the first limitation disallowed such a chance. With the above two limitations listed, it was impossible to put forth a fair comparison of results from *Cobalt* and FLUENT. Instead they had to be separated and analyzed as per events happening aft of the aircraft.

Thirdly, both *Cobalt* and FLUENT grids, when delivered were locked. They did not allow any means of manipulation such as increasing the density of cells, changing the structure of the C-130H models and even adding parachutes and loads analyses. These could have given the research an added dimension, especially with parachutes in place, and the access to grid-independence study. Having a grid-independence study would have greatly

increased the credibility of the solution results. *Cobalt* has the ability of placing an overset grid to be coded onto the existing grid while FLUENT makes use of dynamic meshing. These two abilities enable dynamic motions and if a parachute could be strategically coded on an overset or by dynamic meshing, it would allow a clearer understanding of what a parachute could be experiencing.

It is hopeful still that the information of the unsteady wakes and zones of zero-upsweep influence transpired in this paper would be instrumental in the future designs of military airdrop protocols.

Improvements to future works could include re-building a new C-130H computational model in a format that is *Cobalt* compatible. As the current model was a replica of the actual 1:16 ENSICA model used for wind tunnel purposes, perhaps the re-build could resemble one which DSTA-RSAF would be using for their own test cases. The new model would allow manipulations and additions for a better understanding of the airflow around the empennage region.

Another future endeavour would be to create an overset parachute and load grid for *Cobalt*. Allowing an avenue for the parachute to inflate, move and rotate with six-degrees of freedom would improve the quality of the analyses many fold. Not only would it be for verification purposes but could perhaps even enable the calculation of airdrop precision to a drop spot. That could be another aspect that is crucial to model as it too has a part in the success of airdrop missions.

One other possible future venture aside from improving the numerical setup and simulation would be to look into a re-design of the cargo bay area and cargo hatch to decrease the amount of vortical evolution in flight. It is already known that by extending the tailgate, it reduces the strength and size of the recirculation zone and upsweep felt under the empennage. Further restructuring the shape and size of the cargo bay zone could very well produce a sizeable effect on the turbulence generated at the wake which might give airdrop operations increased opportunities for success.

REFERENCES

- [1] Aircraft History of the C-130. Motoart: History. Updated on 7 Aug 2009,
URL: <http://www.motoart.com/aviation-history/aircraft-history-c-130>.
- [2] C-130 Hercules. The official website of the U.S. Air Force. Updated on 30 Apr 2010, URL: <http://www.af.mil/information/factsheets/factsheet.asp?id=92>
- [3] Farhall, D, 1978. Recovery Systems Designs Guide. Technical Report AFFDL-TR-78-151, Final Report For Period June 1975 to June 1978; Air Force Flight Dynamics Laboratory.
- [4] Seegar, M., Müller, L., Carlsson, P. K., Bury, Y., de Lassat de Pressigny, Y., Lallemand, G., Vallance, M., Wheeler, R., Benney, R., Charles, R., Morton, S. A. & Tavan, S. (2005). Four-Powers Long Term Technology Projects: “Airflow Influence on Airdrop” and “2nd Precision Airdrop Improvements”. *18th AIAA Aerodynamic Decelerator Systems Technology Conference and Seminar*, AIAA 2005-1602.
- [5] Spalart, P. R. & Allmaras, S. R. (1994). A One-Equation Turbulence Model for Aerodynamic Flows. *La Recherche Aerospaciale*, 1(1), 5-21.
- [6] Shur, M. L., Spalart, P. R., Strelets, M. K. & Travin, A. K. (2000). Turbulence Modeling of Rotating and Curved Channels: Assessing the Spalart-Shur Correction, *AIAA Journal*, 38(5), 784-792.

- [7] Versteeg, H. K & Malalasekera, W. (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. 2nd edition, *Pearson Education Limited*. ISBN: 978-0-13-127498-3.
- [8] Menter, F. R.. (1992). Performance of Popular Turbulence Models for Attached and Separated Adverse Pressure Gradient Flows. *AIAA Journal*, 30(8), 2066-2072.
- [9] Menter, F. R. (1993). Zonal Two Equation k- ω Turbulence Models for Aerodynamic Flows. *AIAA Paper*, 93-2906.
- [10] Menter F. R., Kuntz, M. & Langtry, R. (2003) *Ten Years of Industrial Experience with the SST Turbulence Model*. (K. Hanjalic, Y. Nagano, & M. Tummers, Eds.) *Turbulence, Heat and Mass Transfer 4*, 4, 2003. Begell House, Inc.
- [11] Smagorinsky, J. (1963). General Circulation Experiments with the Primitive Equations. *Monthly Weather Review*, 91(3), 99-164.
DOI: 10.1126/science.27.693.594.
- [12] Deardorff, J. W. (1973). The Use of Subgrid Transport Equations in a Three-Dimensional Model of Atmospheric Turbulence. *J. Fluids Eng*, 95, 429-38.
- [13] Deardorff, J. W. (1974). Three-Dimensional Numerical Study of the Height and Mean Structure of a Heated Planetary Boundary Layer. *Boundary-Layer Meteorology*, 7(1), 81-106.
DOI: 10.1007/BF00224974.

- [14] Deardorff, J. W. (1974). Three-Dimensional Numerical Study of Turbulence in an Entraining Mixed Layer. *Boundary-Layer Meteorology*, 7(2), 199-226.
DOI: 10.1007/BF00227913.
- [15] Ferziger, J. H. (1976)/ Large Eddy Simulations of Turbulent flows. *AIAA Journal*, 15, 1261-1267.
- [16] Deardorff, J. W. (1970). A Numerical Study of Three-Dimensional Turbulent Channel Flow at Large Reynolds Numbers. *Journal of Fluid Mechanics*, 41(02), 453-480. Cambridge Univ Press.
DOI: 10.1017/S0022112070000691.
- [17] Lilly, D. K. (1966). On the Application of the Eddy Viscosity Concept in the Inertial Sub-range of Turbulence, *NCAR Manuscript 123*, 123, 21. National Center for Atmospheric Research.
- [18] Rogallo, R. S. & Moin, P. (1984). Numerical Simulation of Turbulent Flows, *Annual Review Fluid Mechanics*, 16(1), 99-137.
- [19] Schumann, U. (1975). Subgrid Scale Model for Finite Difference Simulations of Turbulent Flows in Plane Channels and Annuli. *Journal of Computational Physics*, 18(4), 376-404.
DOI: 10.1016/0021-9991(75)99093-5.

- [20] Leslie, D. C. & Quarini, G. L. (1979). *The Application of Turbulence Theory to the Formulation of Subgrid Modelling Procedures*. *Journal of Fluid Mechanics*, 91, 65-91.
- [21] Mcmillan, O. J., & Ferziger, J. H. (1979). Direct Testing of Subgrid-Scale Models. *AIAA Journal*, 17(12), 1340-1346.
- [22] Clark, R. A., Ferziger, J. H., & Reynolds, W. C. (1979). Evaluation of subgrid-scale models using an accurately simulated turbulent flow. *Journal of Fluid Mechanics*, 91(01), 1-16.
- [23] Bardina, J., Ferziger, J. H., & Reynolds, W. C. (1980). Improved subgrid-scale models for large-eddy simulation. *AIAA*, 80, 1357. American Institute of Aeronautics and Astronautics, Fluid and Plasma Dynamics Conference, 13th, Snowmass, Colo.
- [24] Constantinescu, G. S., & Squires, K. D. (2003). LES and DES Investigations of Turbulent Flow over a Sphere at $Re = 10,000$. *Flow Turbulence and Combustion formerly Applied Scientific Research*, 70(1-4), 267-298.
DOI: 10.1023/B:APPL.00000004937.34078.71.
- [25] Moin, P. (2002). Advances in large eddy simulation methodology for complex flows. *International Journal of Heat and Fluid Flow*, 23(5), 710-720.
DOI: 10.1016/S0142-727X(02)00167-4.

- [26] Mahesh, K., Constantinescu, G. & Moin, P. (2000). Large-eddy Simulation of Gas-Turbine Combustors. *Annual Research Briefs*, Center for Turbulence Research, Stanford Univ.
- [27] Mahesh, K., Constantinescu, G., Apte, S., Iaccarino, G. & Moin, P. (2001). Large-eddy Simulation of Gas-Turbine Combustors. *Annual Research Briefs*, Center of Turbulence Research, Stanford Univ.
- [28] Ghosal, S., & Moin, P. (1995). The Basic Equations for the Large Eddy Simulation of Turbulent Flows in Complex Geometry. *Journal of Computational Physics*. 118, 24-37.
DOI: 10.1006/jcph.1995.1007.
- [29] Vasilyev, O. V., Lund, T. S., & Moin, P. (1998). A General Class of Commutative Filters for LES in Complex Geometries. *Journal of Computational Physics*, 146(1), 82-104.
DOI: 10.1006/jcph.1998.6060
- [30] Spalart, P. R., Jou, W. H., Strelets, M., & Allmaras, S. R. (1997). Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach. In C. Liu & Z. Liu (Eds.), *Advances in DNSLES* (Vol. 1, pp. 4-8). Greyden Press.
- [31] Travin, A., Shur, M., Strelets, M., & Spalart, P. R. (2000). Detached-Eddy Simulations Past a Circular Cylinder. *Combustion*, 63(1), 293-313.
DOI: 10.1115/1.1471532

- [32] Strelets, M. (2001). Detached Eddy Simulation of Massively Separated Flows. *39th AIAA Aerospace Sciences Meeting and Exhibit, 2001-0879*(AIAA-2001-879).
- [33] Spalart, P. R. (2006). *Topics in Detached-Eddy Simulation*. In Computational Fluid Dynamics 2004, 3-12.
- [34] Forsythe, J. R., Squires, K. D., Wurtzler, K. E., & Spalart, P. R. (2002). Detached-Eddy Simulation of Fighter Aircraft at High Alpha Aerospace Sciences Meeting 2002 14-18 January 2002 / Reno, Nevada.
- [35] Squires, K. D., Forsythe, J. R., & Spalart, P. R. (2001). Detached-eddy simulation of the separated flow around a forebody cross-section. (B. J. Geurts, R. Friedrich, & O. Metais, Eds.) *Direct and LargeEddy Simulation IV*, 481500. Kluwer Academic Pub, 481-500.
- [36] Morton, S. A., Forsythe, J. R., Mitchell, A. M., & Hajek, D. (2002). DES and RANS Simulations of Delta Wing Vortical Flows, 40th Aerospace Sciences Meeting & Exhibit Reno, Nevada, AIAA 02-0587.
- [37] Morgan, P. E. (2001). Simulation of Unsteady Three-dimensional Separation on a Pitching Wing 31st AIAA Fluid Dynamics Conference & Exhibit 11-14 June 2001 / Anaheim, CA. A01-31224 AIAA 2001-2709.
- [38] Johnson, W., Trickey, C., Forsythe, J., Albertson, J., Leigh, E. (2002). *Experimental and Computational Investigation of the Flow behind a C-130 with Tailgate Down*. 40th Aerospace Sciences Meeting and Exhibit Reno, Nevada. AIAA 2002-0713.

- [39] Squires, K. D., Forsythe, J. R., Wurtzler, K. E., Strang, W. Z., Tomaro, R. F., Grismer, M. J., Spalart, P. R. (2001). Towards Prediction of Aircraft Spin. High Performance Computing Users Group Meeting.

- [40] Garbaruk, A. V., Shur, M. L., Strelets, M. K., & Travin, A. K. (1994). Supersonic base flow experiments in the near wake of a cylindrical afterbody. (W. Haase, M. Braza, & A. Revell, Eds.) *DESider A European Effort on Hybrid RANSLES Modelling Notes on Numerical Fluid Mechanics and Multidisciplinary Design Vol 103*, 32(1), 77-83.

DOI: 10.2514/3.11953

- [41] Menter, F. R., & Kuntz, M. (2002). Adaptation of Eddy-Viscosity Turbulence Models to Unsteady Separated Flow Behind Vehicles. *The aerodynamics of heavy vehicles trucks buses and trains Volume 1, 2002*, 339-352.

- [42] Spalart, P. R., Deck, S., Shur, M. L., Squires, K. D., Strelets, M. K., & Travin, A. (2006). A New Version of Detached-eddy Simulation, Resistant to Ambiguous Grid Densities. *Theoretical and Computational Fluid Dynamics*, 20(3), 181-195.

DOI:10.1007/s00162-006-0015-0.

- [43] Strang, W. Z., Tomaro, R. F. & Grismer, M. J. (1999). The Defining Methods of Cobalt₆₀: A Parallel, Implicit, Unstructured Euler/Navier-Stokes Flow Solver. 37th AIAA Aerospace Sciences Meeting and Exhibit 11-14 January 1999 Reno, Nevada. A99-16635.

- [44] Cobalt Solutions, LLC Homepage: News. Updated in 2012.

<http://www.cobaltcfd.com/index.php/site/news/>

- [45] Serrano, M., Leigh, E., Johnson III, W., Forsythe, J. R., Morton, S. A. & Squires, K. D. (2003) Computational Aerodynamics of the C-130 in Airdrop Configurations. Aerospace Sciences Meeting 6-9 January 2003 / Reno, Nevada. AIAA 2003-0229.
- [46] Claus, M. P., Morton, S. A., Cummings, R. M. & Bury, Y. (2005). DES Turbulence Modeling on the C-130 Comparison between Computational and Experimental Results. 43rd AIAA Aerospace Sciences Meeting and Exhibit Reno, Nevada, AIAA 2005-884.
- [47] Morton, S. A., Vignes, B. & Bury, Y. (2006). Experimental and Computational Results of a Simplified C-130 Shape Depicting Airflow Influence on Airdrop. RTO-MP-AVT-133.
- [48] Bury, Y., Morton, S. A. & Charles, R. (2008). Experimental Investigation of the Flow Field in the Close Wake of a Simplified C-130 Shape: A Model Approach of Airflow Influence on Airdrop. 26th AIAA Applied Aerodynamics Conference Honolulu, HI.
- [49] Bergeron, K., Cassez, J. F. & Bury, Y. (2009). Computational Investigation of the Upsweep Flow Field for a Simplified C-130 Shape. 47th AIAA Aerospace Sciences Meeting Including The New Horizons Forum and Aerospace Exposition Orlando, FL. AIAA 2009-90.
- [50] GiD Homepage, updated in 2012, URL: <http://gid.cimne.upc.es/>

- [51] ANSYS Fluent webpage, updated in 2012, URL: <http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/ANSYS+Fluent>.
- [52] Pang K. H. S. (2010). Numerical Study of Airdrop Systems From Transport Aircraft. Unpublished Master's preliminary report. Nanyang Technological University.
- [53] Zhong, L. (2005). Effect of RANS Turbulence Models on Computation of Vortical Flows over Wing-Body Configuration. *Trans. Japan Soc. Aero. Space Sci.*, 48(161), 152-160.