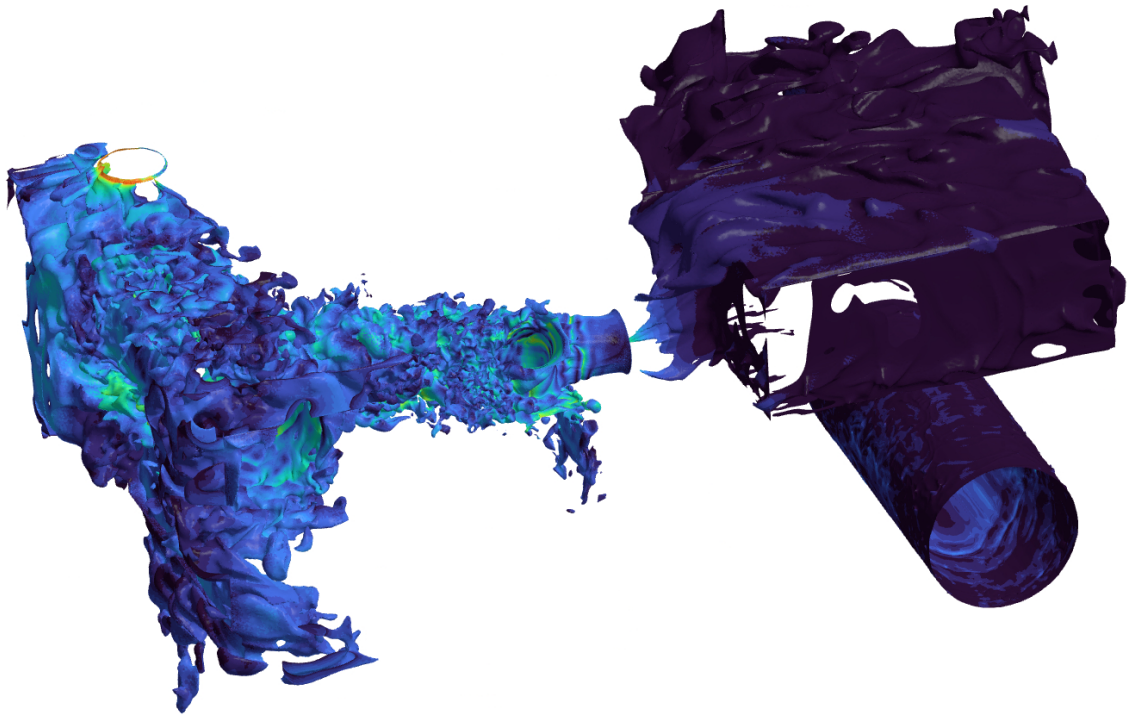




CHALMERS
UNIVERSITY OF TECHNOLOGY



Numerical Methods for Internal Heat Transfer in Airplanes

Master's thesis in Mobility Engineering

Andreas Eriksson

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES

CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026
www.chalmers.se

MASTER'S THESIS 2026

Numerical methods for internal heat transfer in airplanes

Andreas Eriksson



Department of Mechanics and Maritime Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026

Numerical methods for internal heat transfer in airplanes
Andreas Eriksson

© Andreas Eriksson, 2026.

Supervisor: Ester Lundin, SAAB
Supervisor: Johan Nilsson, Faurecia Creo
Examiner: Niklas Andersson, Mechanics and Maritime Sciences

Degree project report 2026
Department of Mechanics and Maritime Sciences
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone +46 31 772 1000

Cover: Vortex visualization of the flow inside an internal bay, created from the isosurface of the Q-criterion and colored to show the velocity magnitude of the flow.

Typeset in L^AT_EX
Gothenburg, Sweden 2026

Numerical methods for internal heat transfer in airplanes
Andreas Eriksson
Department of Mechanics and Maritime Sciences
Chalmers University of Technology

Abstract

Understanding the limits of an airplane is crucial for ensuring the reliability of the airplane and its systems. One limiting factor is the thermal loads from various internal heat sources in the airplane. It is necessary to understand how heat is transported in the internal bays of the airplane to ensure that the airframe and avionics are rated for the thermal environment that is developed. Since the process for designing an airplane is very time-consuming, simulations of the airflow using computational fluid dynamics (CFD) have become the norm in the industry. However, highly resolved turbulent simulations take months or even years to run through. Simplified simulations are often more feasible in industrial applications due to a great reduction in time and computational cost. This thesis explores different numerical methodologies for simplified CFD simulations, evaluating their accuracy and time consumption compared to a highly resolved turbulent simulation. This comparison shows not only which flow structures and flow properties that are captured, but also to what degree. The largest simplification for the simulations in this thesis is the use of the steady state Reynolds-Averaged Navier-Stokes (RANS) solver for all the explored numerical methods. The numerical methods investigated are the turbulence modelling schemes Lag Elliptic Blending (EB) $k - \varepsilon$ and Shear Stress Transport (SST) $k - \omega$, the effects of a polyhedral versus a trimmed cell mesh, the use and number of prism layers for the near-wall regions in the mesh, the effects of two transition models for the SST $k - \omega$ turbulence model, and the effects of the inviscid flux discretization schemes Roe's and Advection Upstream Splitting Method + (AUSM+) Flux-Vector Splitting (FVS).

Keywords: CFD, turbulence, fluid dynamics, aerospace, heat transfer, RANS.

Acknowledgements

First and foremost, I want to express my deepest gratitude to my supervisors Ester Lundin and Johan Nilsson for their continuous guidance and support. I'm glad to have had the opportunity to work with and learn from them both.

I would also like to thank SAAB for the opportunity to undertake this master's thesis and my colleagues at the company. It has been an exciting end to my studies and I will be forever grateful for the time I've spent at the company.

I am immensely grateful to my friends and family for their cheerful support during my studies and this thesis work.

Finally, I would like to thank Niklas Andersson for his work as my examiner at Chalmers during this thesis.

Andreas Eriksson, Gothenburg, July 2026

List of Acronyms

Below is the list of acronyms that have been used throughout this thesis listed in alphabetical order:

AUSM+	Advection Upstream Splitting Method +
BC	Boundary Condition
BL	Boundary Layer
CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy
CHT	Conjugate Heat Transfer
CPU	Central Processing Unit
DES	Detached Eddy Simulations
DNS	Direct Numerical Simulation
EB	Elliptic Blending
FVM	Finite Volume Method
FVS	Flux-Vector Splitting
GCI	Grid Convergence Index
HTLES	Hybrid Temporal Large Eddy Simulations
LES	Large Eddy Simulations
RANS	Reynolds-Averaged Navier-Stokes
RNG	Re-Normalisation Group
SFS	Subfilter-scale
SGS	Subgrid-scale
SRH	Scale Resolving Hybrid
SST	Shear Stress Transport
TPITM	Temporal Partially Integrated Transport Model

Nomenclature

Below is the nomenclature of indices, parameters, and variables that have been used throughout this thesis.

Indices

f	Indice for cell face
i,j,k	Indices for tensor notation
n	Indice for size of mesh

Variables

α	Blending factor
γ	Intermittency
Γ	Preconditioning matrix in Roe's inviscid flux discretization scheme
Δ	Local length-scale
ΔQ_{ij}	Reconstructed solution vector in Roe's inviscid flux discretization scheme
Δt	Timestep
Δx	Filter size
δ_{ij}	Kronecker delta for tensor index i and j
ε	Dissipation rate
ϵ	Error of a flow variable between two meshes
η	Timescale ratio
θ	Temperature
κ	Kolmogorov scale
λ	Taylor microscale
μ	Dynamic viscosity

μ_t	Turbulent dynamic viscosity
ν	Kinematic viscosity
ν_t	Turbulent kinematic viscosity
ρ	Density
τ_{ij}	Shear stress tensor
τ_{sfs}	Subfilter-scale shear stress tensor
Υ	Compressibility modification
Φ_{ij}	Dissipation term
φ	Lag parameter
ϕ	General flow variable
ω	Specific dissipation rate
a_{ij}	Anisotropy tensor
c_p	Specific heat capacity
D_ω	Cross-diffusion term
E_γ	Destruction term for intermittency
e	Relative error
F_1	Blending function
F_2	Blending function
f_μ	Dampening function
f_c	Curvature correction factor
f_f	Inviscid flux at a face
G_ω	Specific dissipation rate production term
G_b	Buoyancy production term
G_k	Turbulent production term
G_{nl}	Non-linear production term
g_i	Gravitational acceleration vector
HT	Heat transfer
h	Representative mesh size
h_{cell}	Cell size
h_{htc}	Heat transfer coefficient
k	Turbulent kinetic energy
k_m	Modelled turbulent kinetic energy
k_r	Resolved turbulent kinetic energy
k_t	Total turbulent kinetic energy

k_{sfs}	Subfilter-scale turbulent kinetic energy
l_η	Kolmogorov length scale
$\dot{m}$	Massflow
N	Number of cells in the mesh
N_n	Mesh size of the n :th mesh
P	Power
P_γ	Production term for intermittency
P_ε	Production term for dissipation rate
P_φ	Production term for lag parameter
P_ω	Production term for specific dissipation rate
P_k	Production term for turbulent kinetic energy
p	Pressure
p_{ij}	Pressure part of the inviscid flux in Liou's AUSM+ inviscid flux discretization scheme
p_{rel}	Relative pressure
Pr_t	Turbulent Prandtl number
R_{ij}	Reynolds stress tensor
R_k	Thermal resistance
Re	Reynolds number
Re_θ	Momentum thickness Reynolds number
$Re_{\theta t}$	Transition momentum thickness Reynolds number
r_k	Theoretical energy ratio
s_{ij}	Strain-rate tensor
T	Turbulent time scale
t	Time
U_s	Sweeping velocity
V_{cell}	Volume of the cell
V_{fluid}	Volume of the fluid
v_i	Velocity vector
w_{ij}	Vorticity tensor

Contents

List of Acronyms	ix
Nomenclature	xi
List of Figures	xvii
List of Tables	xxiii
1 Introduction	1
1.1 Background	1
1.2 Previous work	2
1.3 Purpose	2
1.4 Goal	3
1.5 Limitations / Demarcations	3
2 Theory	5
2.1 Fluid dynamics and CFD	5
2.1.1 Reynolds-Averaged Navier-Stokes simulations	6
2.1.2 Large Eddy Simulations	6
2.1.3 Scale Resolving Hybrid simulations	7
2.2 Turbulence models	9
2.2.1 SST k-omega	9
2.2.2 Lag EB k-epsilon	12
2.3 Spatial discretization	15
2.3.1 Mesh convergence study through GCI	17
2.3.2 Cell size for highly resolved turbulence	18
2.3.3 Transition modelling	19
2.4 Inviscid flux discretization schemes	20
3 Method	21
3.1 Problem description	21
3.2 Geometry	21
3.3 Boundary conditions	23
3.4 RANS simulations	24
3.4.1 Simulation setup	24
3.4.2 Spatial discretization	25
3.5 SRH simulation	26

3.5.1	Initial conditions	26
3.5.2	Simulation setup	26
3.5.3	Spatial discretization	27
3.5.4	Temporal discretization	29
4	Results and Discussion	31
4.1	SRH mesh resolution	31
4.2	RANS mesh convergence study	35
4.3	Simulation results	38
4.3.1	SRH results	38
4.3.2	Numerical comparison of the RANS simulations	41
4.3.3	Comparison of the fluid fields	44
4.3.4	Comparison of velocity profiles	63
4.3.5	Comparison of surface temperatures	65
4.4	Resource consumption	80
5	Conclusion	81
5.1	Future work	82
	Bibliography	83

List of Figures

2.1	The three common meshing methods with the same settings used to discretize the same block.	16
2.2	Energy spectrum with Taylor microscale and Kolmogorov scale marked. The Kolmogorov law $-5/3$ decay is also shown in this figure. Note that the Taylor microscale is placed in the inertial subrange.	19
2.3	A representation of the boundary layer along a flat surface where the different regions are shown.	19
3.1	Three separate views of the fluid geometry with the objects in the domain colored. The aft wall is at the highest value on the x -axis and the forward wall is at the lowest value on the x -axis. The blue block in the aft bay represents avionics 1. The orange block in the forward bay represents avionics 2, the blue block in the forward bay represents avionics 3. The dark green pipe in the forward bay represents the avionics pipe. The yellow pipe in both the aft and forward bay represents an isolated pipe. The light green pipe in the aft bay represents a pipe with a leaking valve. The red circle on top of the aft bay and the cyan circle on the bottom of the forward bay are the Outlet and Inlet, respectively. The circled region on the pipe with the leaking valve in the aft bay in Figure 3.1c shows the location of the leaking valve.	22
3.2	The wall Y^+ of the SRH simulation. The regions where the wall Y^+ are above one are located at impingement zones, the ventilation hole, or close to the inlet, leakage and outlet boundary conditions. . .	27
3.3	The three locations used for evaluating the resolution of the simulation for SRH. The blue arrow represents the flow through the ventilation hole between the bays.	28
3.4	Figure showing the Convective Courant number (CFL) in the domain. The regions where cells with a CFL number larger than 1 are circled in this figure.	29
3.5	Plots showing the behaviour of the residuals for the SRH simulation as well as the behaviour of the velocity in a point in the domain. . . .	30
4.1	Two-point correlations of the fluctuating velocity components in turbulent structures residing at location 1 in the domain.	32
4.2	Two-point correlations of the fluctuating velocity components in turbulent structures residing at location 2 in the domain.	32

4.3	Two-point correlations of the fluctuating velocity components in turbulent structures residing at location 3 in the domain.	33
4.4	Energy spectra based on the autocorrelation of the fluctuating velocity at location 1. The $-5/3$ decay based on the Kolmogorov law is plotted as the dashed line.	33
4.5	Energy spectra based on the autocorrelation of the fluctuating velocity at location 2. The $-5/3$ decay based on the Kolmogorov law is plotted as the dashed line.	34
4.6	Energy spectra based on the autocorrelation of the fluctuating velocity at location 3. The $-5/3$ decay based on the Kolmogorov law is plotted as the dashed line.	34
4.7	A cross-sectional view of the shielding function to visualize where LES is active and where RANS is active. Pure LES mode is activated when the shielding function is 1. Pure RANS mode is activated when the shielding function is 0. When the shielding function is between 0 and 1, the SRH simulation blends the two simulation modes. The location of the cross-section is shown in the top right corner of the figure.	35
4.8	The convergence of the surface-averaged temperature at the outlet for the different mesh sizes used for the polyhedral and trimmed cell meshes.	37
4.9	A cross-sectional view of the instantaneous relative pressure field in the bay.	38
4.10	A cross-sectional view of the instantaneous velocity field of the SRH simulation.	39
4.11	A cross-sectional view of the time-averaged velocity field of the SRH simulation.	39
4.12	A cross-sectional view of the instantaneous temperature of the SRH simulation.	40
4.13	A cross-sectional view of the time-averaged temperature of the SRH simulation.	41
4.14	Wall Y^+ values of the reference simulation. Only wall Y^+ values above 1 are shown in the figure.	44
4.15	Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.	44
4.16	Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.	45
4.17	Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.	45
4.18	Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.	45
4.19	Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.	46
4.20	Mean velocity field of the reference simulation shown at the right cross-section.	47

4.21	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.	47
4.22	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.	48
4.23	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.	48
4.24	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.	48
4.25	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.	49
4.26	Mean temperature field of the reference simulation shown at the right cross-section.	50
4.27	Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.	50
4.28	Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.	51
4.29	Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.	51
4.30	Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.	52
4.31	Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.	52
4.32	Mean velocity field of the reference simulation shown at the top mid- dle cross-section.	54
4.33	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.	54
4.34	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.	55
4.35	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.	55
4.36	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.	55

4.37	Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.	56
4.38	Mean temperature field of the reference simulation shown at the top middle cross-section, the location of which is shown in the top right of the figure.	57
4.39	Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38. . .	57
4.40	Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38. . .	58
4.41	Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38. . .	58
4.42	Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38. . .	58
4.43	Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38. . .	59
4.44	Isometric view of the isosurface of the temperature at 200 °C from the reference simulation.	60
4.45	Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.	60
4.46	Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.	61
4.47	Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.	61
4.48	Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.	62
4.49	Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.	62
4.50	Figures showing the locations of the measured data presented in Figure 4.51.	64
4.51	Figure showing the velocity profiles of the simulations along the top surface of avionics 1. The figure only shows the profile of the x -component of the velocity, since this is the dominating component due to the jet.	64
4.52	Figures showing the locations of the measured data presented in Figure 4.53.	65
4.53	Figure showing the velocity profiles of the simulations along the top surface of avionics 1. The figure only shows the profile of the x -component of the velocity, since this is the dominating component due to the jet.	65
4.54	Reference simulation showing the temperature of the top surface of avionics 1. The location of the plane is shown highlighted in the top right of the figure.	66
4.55	RANS simulations showing the temperature of the top surface of avionics 1.	67

4.56	RANS simulations showing the temperature of the top surface of avionics 1.	67
4.57	RANS simulations showing the temperature of the top surface of avionics 1.	68
4.58	RANS simulations showing the temperature of the top surface of avionics 1.	68
4.59	RANS simulations showing the temperature of the top surface of avionics 1.	69
4.60	Reference simulation showing the temperature of the forward surface of avionics 1. The location of the plane is shown highlighted in the top right of the figure.	70
4.61	RANS simulations showing the temperature of the forward surface of avionics 1.	71
4.62	RANS simulations showing the temperature of the forward surface of avionics 1.	71
4.63	RANS simulations showing the temperature of the forward surface of avionics 1.	72
4.64	RANS simulations showing the temperature of the forward surface of avionics 1.	72
4.65	RANS simulations showing the temperature of the forward surface of avionics 1.	73
4.66	Reference simulation showing the temperature of the front surface of the isolated pipe in the aft bay. The location of the surface is shown highlighted in the top right of the figure.	74
4.67	RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.	74
4.68	RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.	74
4.69	RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.	75
4.70	RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.	75
4.71	RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.	75
4.72	Reference simulation showing the temperature of the bottom surface of avionics 2. The location of the surface is shown highlighted in the top right of the figure.	76
4.73	RANS simulations showing the temperature of the bottom surface of avionics 2.	77
4.74	RANS simulations showing the temperature of the bottom surface of avionics 2.	77
4.75	RANS simulations showing the temperature of the bottom surface of avionics 2.	78
4.76	RANS simulations showing the temperature of the bottom surface of avionics 2.	78

4.77 RANS simulations showing the temperature of the bottom surface of avionics 2.	79
---	----

List of Tables

2.1	Model coefficients for the SRH solver.	9
2.2	Table showing the constant values for the model coefficients in the SST $k - \omega$ turbulence model.	12
2.3	Model coefficients for the Lag EB $k - \varepsilon$ turbulence model. In $C_{\varepsilon 3}$, $v_{\parallel}$ and $v_{\perp}$ are the parallel and perpendicular velocity components to the gravitational vector g_i	15
3.1	Boundary conditions for all the simulations in this work.	23
3.2	Table showing the general setup of each RANS simulation.	24
3.3	The initial conditions for the steady RANS simulations.	24
3.4	The solver settings used for the RANS simulations in StarCCM+.	25
3.5	The mesh size and its representative mesh size used in the mesh convergence study of the polyhedral meshing method.	25
3.6	Mesh refinement factor and the related meshes for the polyhedral meshing method.	25
3.7	The mesh size and its representative mesh size used in the mesh convergence study of the trimmed meshing method.	25
3.8	Mesh refinement factor and the related meshes for the trimmed meshing method.	26
3.9	The solver settings used for the SRH simulation in StarCCM+.	27
4.1	Results from GCI analysis of surface-averaged heat transfer and mean temperature for three different polyhedral meshes of the domain. The variables $\bar{\theta}_{outlet}$ and $\bar{\theta}_{fluid}$ denote the surface-averaged mean temperature of the outlet and volume-averaged mean temperature of the fluid, respectively.	36
4.2	Results from GCI analysis of surface-averaged heat transfer and mean temperature for three different trimmed meshes of the domain.	36
4.3	Surface-averaged mean temperature of the parts inside the bay and of the outlet for the SRH simulation. The RANS simulation values are the error compared to the reference SRH simulation, where a negative error implies that the result was lower and a positive error implies that the result was higher than the reference.	42

4.4	Mean heat transfer from the fluid to the different parts inside the bay and to the outlet of the bay of the SRH simulation. The values for the RANS simulations are shown as errors compared to the reference simulation, where a negative error implies that the result was lower and a positive error implies that the result was higher than the reference.	43
4.5	The resource consumption for each simulation.	80

1

Introduction

1.1 Background

Airplanes are subjected to many heat sources, both internal and external. These heat sources generate thermal loads on the airplane's structure and components. Therefore, there is a great need to understand what temperatures will occur in different parts of the airplane and how the heat spreads. The internal bays in airplanes are filled with avionics and structural components. Due to the geometry of the bays, the flow fields inside the internal bays can become quite complex. Understanding the flow field and what temperatures will emerge in the internal bays early on in the design process is crucial, as design changes become more costly and time-consuming the further into the design process they are implemented.

The flow fields are often analyzed using conjugate heat transfer (CHT) in computational fluid dynamics (CFD) simulations, where the heat transfer in solids is simulated together with a flow and transport of heat in a fluid. The most expensive part of these simulations is solving the convective heat transfer in turbulent flows; turbulent simulations are expensive since turbulence is chaotic by nature. Simulations that provide highly resolved turbulence solutions, like Large Eddy Simulations (LES) or hybrid Reynolds-Averaged Navier-Stokes (RANS)-LES methods, are expensive and take a long time in comparison to standard steady-state RANS methods. However, the simplifications that steady-state RANS methods entail might introduce large errors in the solution since the methods do not account for fluctuations in the flow. Highly resolved turbulent flow simulations can be too expensive to be practical in industrial cases, so a compromise is made to use steady-state RANS simulations. With this compromise, it is important to understand which flow structures and flow properties that are captured and to what degree. There are a number of options when it comes to setting up steady-state RANS simulations. The optimal settings are highly dependent on the fluid geometry and properties; some settings work better for one type of flow, while other flows need completely different settings to get a reasonable result.

1.2 Previous work

A number of thermal studies on confined spaces in aircraft have previously been conducted, with applications such as airplanes and helicopters. Geron et al. [1] investigated the accuracy of RANS simulations of a crown compartment of a passenger airplane (the upper compartment between the passenger cabin and the fuselage) using a Re-Normalisation Group (RNG) $k - \varepsilon$ turbulence model with a surface-to-surface radiation model. The RANS simulations were validated with experimental data from a model of a crown compartment and the results from the simulations agreed well with the experimental results. The fluid field of a rotorcraft avionics bay with active cooling was investigated by Akin and Kahveci [2], with the goal of optimizing the inlet and outlet positions for cooling air. The optimized locations were validated using a coupled CFD solver with the realizable $k - \varepsilon$ turbulence model. Butler and Newport [3] studied how the placement of avionics affects the flow field inside a crown compartment of a passenger airplane. They found that the flow field in the compartment was highly dependent on the jet of cold air from the inlet, the heating from the avionics device and the thermal stratification of the fuselage. Butler and Newport state that similar types of analysis should be undertaken for other confined areas of the aircraft, as similar complex flow fields are expected.

1.3 Purpose

An internal bay in an aircraft is tightly packed with avionics and structural components that interfere with the incoming cooling airflow inside these bays, creating a complex flow field. Highly resolved turbulent simulations can be used to accurately capture the effects of turbulence in such a complex flow, but they require a lot of time and resources. Reducing the computational cost for accurate simulations of the flow is therefore important for the aerospace industry. The objective of this work is to evaluate numerical modelling methods used to predict convective heat transfer within the internal bays of an aircraft. This will be achieved through studying simplified CFD simulations and their effects on the convective heat transfer and temperature of the components inside the bay, comparing their results to those of a highly resolved hybrid RANS-LES simulation. No study of how well the flow phenomena are captured and to what degree when using steady-state RANS models to solve the flow has previously been made at SAAB, which is why it is studied in this thesis.

1.4 Goal

The goal of this thesis is to find a simulation methodology that results in accurate steady-state RANS simulations for complex geometries. To reach this goal, the following research questions will be investigated:

1. What effects does the choice of turbulence model (SST $k - \omega$ or Lag EB $k - \varepsilon$) have on the predicted convective heat transfer and temperature of the boundaries?
2. How does the prism layer affect the predicted convective heat transfer and temperature of the boundaries?
3. How does the convective heat transfer and temperature differ between a polyhedral mesh and a trimmed mesh of similar size?
4. What is the computational cost of the different simulation settings?
5. How does the predicted convective heat transfer and temperature of the simplified simulations compare to that of the highly resolved simulation?

1.5 Limitations / Demarcations

Conductive heat transfer in solids is not included in this study. This means that the heat will not spread through the walls of the objects inside the bay nor through the walls of the bay itself. The limitation implies that simulations of the heat transfer in a solid will not be carried out.

Radiative heat transfer, the heat transfer due to electromagnetic waves emitted by an object, will also be excluded in this study.

2

Theory

In this chapter, the theory behind the simulations is discussed. All the equations in this chapter are written using tensor notation. Simcenter Star-CCM+ is the CFD solver software used in this thesis and all the equations are therefore written in relation to this specific solver.

2.1 Fluid dynamics and CFD

Fluids make up a major part of the world we live in. The manipulation of fluids is the key to our technological development as a species, especially fluid dynamics, the study of fluids in motion. Knowledge in this field is used in everything from power generation and transportation to medicine and heating. A phenomenon that occurs in fluid dynamics is turbulence. Turbulence is a chaotic phenomenon that enhances the mixing of fluids and thus the heat transfer in fluids. Solving the fluid dynamics equations by hand requires a lot of time, but with the introduction of computers and computational fluid dynamics (CFD) simulation software, the calculation time was significantly reduced and the task of calculating flow by hand was eliminated.

Computational fluid dynamics simulations are used to approximate the motions and properties of fluids. The simulations are based on solving the continuity equation, the Navier-Stokes equations of motion and the energy transport equation. The aforementioned equations are shown using tensor notation [4] in equations 2.1, 2.2 & 2.3 respectively. Equations 2.4 and 2.5 show the definitions of the shear-stress tensor τ_{ij} and the strain-rate tensor s_{ij} , respectively. The dissipation term Φ_{ij} is defined as in equation 2.6.

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho v_i}{\partial x_i} = 0 \quad (2.1)$$

$$\frac{\partial \rho v_i}{\partial t} + \frac{\partial \rho v_i v_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (2.2)$$

$$\frac{\partial \rho c_p \theta}{\partial t} + \frac{\partial \rho v_i c_p \theta}{\partial x_i} = -\frac{\partial p v_i}{\partial x_i} + \Phi_{ij} + \frac{\partial}{\partial x_i} \left(k \frac{\partial \theta}{\partial x_i} \right) \quad (2.3)$$

$$\tau_{ij} = 2\mu s_{ij} - \frac{2}{3}\mu \frac{\partial v_k}{\partial x_k} \delta_{ij} \quad (2.4)$$

$$s_{ij} = \frac{1}{2} \left(\frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) \quad (2.5)$$

$$\Phi_{ij} = 2\mu \left(s_{ij} - \frac{1}{3} s_{kk} \delta_{ij} \right)^2 \quad (2.6)$$

The approach to represent the fluid domain in CFD by discretizing it using the Finite Volume Method (FVM), where the domain is split into small cells, is used in this work. The continuity, Navier-Stokes and energy equations are computed for each cell, with a dependence on the neighboring cells at the previous iteration. For a steady-state simulation, a converged solution will be reached eventually through an iterative process of computations for the mesh of cells over the entire domain. If the simulation is time-dependent (unsteady), an instantaneous solution will exist for each timestep.

2.1.1 Reynolds-Averaged Navier-Stokes simulations

The idea behind steady Reynolds-Averaged Navier-Stokes (RANS) simulations is to time-average and solve the Navier-Stokes equations (equations 2.1 & 2.2), making the solution independent of time. The space x and time t dependent flow variables $\phi(x, t)$ can be decomposed into a time-averaged part that is only dependent on space $\langle \phi \rangle(x)$ and a fluctuating part $\phi'(x, t)$, dependent on both time and space, as equation 2.7 shows. The definition of the time-averaged part is shown in equation 2.8 [5]. Solving the time-averaged Navier-Stokes equations numerically is problematic since the system of equations contains four equations, one continuity equation 2.1 and three momentum equations 2.2, but contains six unknown Reynolds stress variables, R_{ij} defined as equation 2.9, resulting in a closure problem. The method used to close the problem in this work is to model the Reynolds stress tensor in its entirety through turbulence models [5]. The turbulence models used in this work are mentioned in detail in Section 2.2 below.

$$\phi(x, t) = \langle \phi \rangle(x) + \phi'(x, t) \quad (2.7)$$

$$\langle \phi \rangle(x) = \frac{1}{2T} \int_{-T}^T \phi(x, t) dt \quad (2.8)$$

$$R_{ij} = -\rho \langle v'_i v'_j \rangle \quad (2.9)$$

2.1.2 Large Eddy Simulations

The main idea behind Large Eddy Simulations (LES) is to resolve the large turbulent eddies and model the small turbulent eddies. In LES, spatial filtering (volume averaging) is employed to separate the larger eddies from the smaller eddies. The filter captures the larger eddies and together with a subgrid-scale (SGS) model that models the smaller eddies, the flow can be simulated. A model for the unresolved SGS stresses is used to represent the interaction between the larger eddies and the smaller eddies in LES, since the interactions are not captured in the filtering [6]. The filtered variables are still dependent on both space and time, compared to the time-averaged variables in RANS, which are only dependent on space. LES is more

accurate than RANS for simulating turbulence, but it is also significantly more computationally demanding than RANS, since it resolves a part of the turbulence instead of modelling all the turbulence. The mesh resolution requirements for accurate LES simulations are very high, especially in the near-wall region [7]. This mesh resolution requirement entails that the cells in the mesh become very small and therefore the simulation timestep is also required to be very small, so as not to have a Courant-Friedrichs-Lewy (CFL) number far greater than 1. These resource issues make LES practically unusable in most industrial cases, due to the limitations of hardware and time. The filtering of the generalized flow variable ϕ is done according to equation 2.10. The Reynolds decomposition of ϕ is shown in equation 2.11, where $\bar{\phi}$ is the filtered variable and ϕ'' is the sub-grid variable. The variables x and Δx denote space and filter size, respectively. The filtered equations are obtained by inserting the decomposed variables $\bar{\phi}$ and ϕ'' into equations 2.1 & 2.2.

$$\bar{\phi}(x, t) = \frac{1}{\Delta x} \int_{x-0.5\Delta x}^{x+0.5\Delta x} \phi(x, t) dx \quad (2.10)$$

$$\phi(x, t) = \bar{\phi}(x, t) + \phi''(x, t) \quad (2.11)$$

2.1.3 Scale Resolving Hybrid simulations

Hybrid RANS-LES models are a combination of the RANS and LES models. RANS is used to model the flow close to the wall and LES is used to resolve the bulk flow. The regions where the models are active is determined by a shielding function f_s , which goes to 0 close to the wall, imposing RANS, and 1 far away from the wall, imposing LES. The main purpose of hybrid RANS-LES methods is to capture the interesting flow regions using LES and model the rest of the domain using RANS. Using this hybrid methodology, interesting regions in the bulk flow can be resolved well without the high computational cost of LES across the entire domain. This is advantageous near walls, where the mesh resolution demands for LES are typically very high [8]. The Scale Resolving Hybrid (SRH) model that is implemented in StarCCM+ is a continuous hybrid temporal RANS-LES model based on the consistent continuous hybrid temporal LES (HTLES) approach developed by Duffal et al. [8]. The SRH model is derived from the Temporal Partially Integrated Transport Model (TPITM). The equations for the SRH model that are implemented in StarCCM+ are obtained by a spatial-temporal filtering [7], where each solution variable is decomposed into a filtered and a subfiltered part, as shown in equation 2.12.

$$\phi = \bar{\phi} + \phi' \quad (2.12)$$

Where:

$$\begin{aligned} \phi &: \text{Solution variable} \\ \bar{\phi} &: \text{Filtered value} \\ \phi' &: \text{Subfiltered value} \end{aligned}$$

The transport equations are produced by inserting the decomposed variables into the Navier-Stokes equations, with the subfilter scale (SFS) shear stress tensor, τ_{sfs} . The SFS shear stress tensor is modelled using the Boussinesq approximation with the SFS kinetic energy, k_{sfs} , as shown in equation 2.13. μ_t is the turbulent viscosity which is calculated as equation 2.28 in Section 2.2.1 and δ_{ij} is the Kronecker delta.

$$\tau_{sfs} = -2\mu_t s_{ij} + \frac{2}{3}\rho k_{sfs} \delta_{ij} \quad (2.13)$$

The transport equation for the SFS turbulent kinetic energy k_{sfs} is based on the SST $k - \omega$ transport equation for k and is calculated as equation 2.14. ψ_H is a parameter calculated according to equation 2.15 using the shielding function f_s , the theoretical energy ratio r_k , the total kinetic energy k_t and the subfilter scale kinetic energy k_{sfs} . The shielding function f_s is calculated according to equation 2.16, where ν , ε and d are the kinematic viscosity, dissipation rate and wall distance, respectively. The shielding function becomes zero close to the walls and is used to force the solver to switch to RANS mode in the region where the mesh resolution is too coarse for LES. r_{dc} is a model coefficient found in Table 2.1. The total kinetic energy is calculated from the modelled and resolved kinetic energies as equations 2.17-2.19. $P_{k_{sfs}}$ is calculated in the same way as P_k for the SST $k - \omega$ model in equation 2.26 in Section 2.2.1 and the terms k_0 and ω_0 are the sustaining terms that counteracts turbulent decay.

$$\frac{\partial \rho k_{sfs}}{\partial t} + \frac{\partial \rho k_{sfs} \bar{v}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k_{sfs}}{\partial x_j} \right] + P_{k_{sfs}} - \rho \beta^* (\psi_H \omega k_{sfs} - \omega_0 k_0) \quad (2.14)$$

$$\psi_H = 1 - f_s + f_s \frac{1}{r_k} \left(\frac{4}{3} - \frac{1}{3} r_k^{3/4} \right) \frac{k_{sfs}}{k_t} \quad (2.15)$$

$$f_s = 1 - \tanh \left[\left(r_{dc} \frac{\nu^{3/4}}{\varepsilon^{1/4} d} \right)^8 \right] \quad (2.16)$$

$$k_m = \overline{k_{sfs}} \quad (2.17)$$

$$k_r = \frac{1}{2} (\bar{\tilde{v}_i \tilde{v}_i} - \bar{\tilde{v}_i} \bar{\tilde{v}_i}) \quad (2.18)$$

$$k_t = k_m + k_r \quad (2.19)$$

The theoretical energy ratio r_k is calculated as equation 2.20 below, where CFL_H (equation 2.21) is the Courant Friedrichs Levy (CFL) number of the filtered quantities. Δt and Δ are the time-step and the local length-scale. The sweeping velocity U_s in equation 2.21 is calculated according to equation 2.22.

$$r_k = 1 - \max\left(0, 1 - \beta_\psi \text{CFL}_H^{2/3}\right) \quad (2.20)$$

$$\text{CFL}_H = \frac{\varepsilon}{k_t^{3/2}} \max(\Delta t U_s, \Delta) \quad (2.21)$$

$$U_s = |\tilde{v}_j| + \sqrt{k_t} \quad (2.22)$$

The shielding function f_s is also used in the definition of the blending factor σ_{HU} , equation 2.23, for the hybrid upwind differencing scheme for the convective flux. The model coefficients for the SRH solver are found in Table 2.1 below.

$$\sigma_{HU} = 1 - f_s \quad (2.23)$$

Table 2.1: Model coefficients for the SRH solver.

Coefficient	Value
β^*	0.09
β_ψ	0.69
σ_k	Eq. 2.43
r_{dc}	45

2.2 Turbulence models

Modelling of the Reynolds stress tensor R_{ij} is often done using turbulence models to close the continuity, Navier-Stokes and energy transport equations. The turbulence models that are relevant for the numerical analysis are described in this section. The equations are taken from the StarCCM+ User Guide [7] as that is the software that is used in this work.

2.2.1 SST k - ω

The Shear Stress Transport (SST) $k - \omega$ turbulence model is a well-established two-equation model that has proven to be reliable in flows with adverse pressure gradient and separations, like the flow over an aircraft wing. The model does, however, have issues predicting turbulence in stagnation regions and accelerating flows [9]. The $k - \omega$ model works well in near-wall regions but struggles with free-stream flows, whereas the $k - \varepsilon$ model works well with free-stream flows but struggles with adverse pressure gradient flows and turbulent boundary layers [9], [10]. The SST $k - \omega$ model combines the $k - \omega$ and $k - \varepsilon$ models, minimizing the main weaknesses of the two models. A blending function is used in the SST $k - \omega$ model to get the effects

of the $k - \omega$ model in the inner boundary layer (BL) closest to the wall, and the $k - \varepsilon$ model in the outer BL of the near-wall flow [5]. The SST $k - \omega$ model that is implemented in StarCCM+ has been modified to include nonlinear constitutive relations to account for anisotropic turbulence [7]. The equations shown in this chapter contain the quadratic constitutive relation since it is used in this work.

Equations 2.24 and 2.25 are the transport equations for the turbulent kinetic energy k and the specific dissipation rate ω , respectively. The transport equations look similar to the transport equations in the $k - \varepsilon$ model but the production terms P_k and P_ω in the $k - \omega$ transport equations have been expanded to include a non-linear production term G_{nl} in the transport equation for k and a cross-diffusion term D_ω in the transport equation for ω (equations 2.26 & 2.27). k_0 and ω_0 are sustaining terms defined from ambient turbulence values that counteract turbulence decay. In free-stream flow, these terms will cancel the destruction terms perfectly. Spalart et al. [11] found this approach of using sustaining terms better than the approach of imposing floor values for limiting the turbulent decay.

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k \bar{v}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \beta^* (\omega k - \omega_0 k_0) \quad (2.24)$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial \rho \omega \bar{v}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + P_\omega - \rho \beta (\omega^2 - \omega_0) \quad (2.25)$$

$$P_k = G_k + G_{nl} \quad (2.26)$$

$$P_\omega = G_\omega + D_\omega \quad (2.27)$$

The turbulent dynamic viscosity for the SST $k - \omega$ turbulence model with Durbin Scale Limiter used in this thesis is described in equation 2.28 where the turbulent timescale T is calculated using equation 2.29. The Durbin Scale Limiter was developed by Durbin [12] to reduce the overprediction of turbulent kinetic energy in stagnation point flows. The mean strain rate tensor is calculated as equation 2.30, the model coefficient a^* is calculated as equation 2.31 and the blending functions F_1 & F_2 are computed as equations 2.32 and 2.34 respectively. $CD_{k\omega}$ is the cross-diffusion coefficient in the F_1 blending function and is defined as equation 2.33. The other model coefficients are presented in Table 2.2.

$$\mu_t = \rho k T \quad (2.28)$$

$$T = \min \left(\frac{1}{\max(\frac{\omega}{a^*}, \frac{\bar{s} F_2}{a_1})}, \frac{C_T}{\sqrt{3\bar{s}}} \right) \quad (2.29)$$

$$\bar{s} = \sqrt{2s_{ij}s_{ij}} \quad (2.30)$$

$$a^* = F_1 + (1 - F_1) \quad (2.31)$$

$$F_1 = \tanh\left(\min\left[\max\left[\frac{\sqrt{k}}{\beta^*\omega d}, \frac{500\nu}{d^2\omega}\right], \frac{2k}{CD_{k\omega}d^2}\right]^4\right) \quad (2.32)$$

$$CD_{k\omega} = \max\left[\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i}, 10^{-20}\right] \quad (2.33)$$

$$F_2 = \tanh\left(\left(\max\left(\frac{2\sqrt{k}}{\beta^*\omega d}, \frac{500\nu}{d^2\omega}\right)\right)^2\right) \quad (2.34)$$

The production terms in the transport equation for kinetic turbulent energy (2.24) in the SST $k - \omega$ model are shown in equations 2.35 and 2.36, where G_k is the term for turbulent production and G_{nl} is the term for non-linear production. O_{ij} in the equation for nonlinear production is calculated as 2.37 where w_{ij} is the vorticity tensor. The term f_c is the curvature correction factor.

$$G_k = \mu_t f_c \bar{s}^2 - \frac{2}{3}\rho k \frac{\partial \bar{v}_j}{\partial x_j} - \frac{2}{3}\mu_t \left(\frac{\partial \bar{v}_j}{\partial x_j}\right)^2 \quad (2.35)$$

$$G_{nl} = -2\mu_t 0.04645 (O_{ij}s_{jk} - s_{ij}O_{jk}) \frac{\partial \bar{v}_i}{\partial x_j} \quad (2.36)$$

$$O_{ij} = \frac{w_{ij}}{\sqrt{(s_{ij} - w_{ij})(s_{ij} - w_{ij})}} \quad (2.37)$$

The production terms in the transport equation for the specific dissipation (2.25) in the SST $k - \omega$ model are the specific dissipation term G_ω and the cross-diffusion term D_ω . The coefficient γ is calculated using coefficients γ_1 and γ_2 with the blending function F_1 as is shown in equations 2.39 - 2.41.

$$G_\omega = \rho\gamma \left[\left(\bar{s}^2 - \frac{2}{3} \left(\frac{\partial \bar{v}_i}{\partial x_i} \right)^2 \right) - \frac{2}{3} \omega \frac{\partial \bar{v}_i}{\partial x_i} \right] \quad (2.38)$$

$$\gamma = F_1\gamma_1 + (1 - F_1)\gamma_2 \quad (2.39)$$

$$\gamma_1 = \frac{\beta_1}{\beta^*} - \sigma_{\omega 1} \frac{\kappa^2}{\sqrt{\beta^*}} \quad (2.40)$$

$$\gamma_2 = \frac{\beta_2}{\beta^*} - \sigma_{\omega 2} \frac{\kappa^2}{\sqrt{\beta^*}} \quad (2.41)$$

$$D_\omega = 2\rho(1 - F_1)\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \bar{v}_i}{\partial x_j} \quad (2.42)$$

The coefficients σ_k and σ_ω are calculated using the blending function F_1 according to equations 2.43 and 2.44.

$$\sigma_k = F_1\sigma_{k1} + (1 - F_1)\sigma_{k2} \quad (2.43)$$

$$\sigma_\omega = F_1\sigma_{\omega 1} + (1 - F_1)\sigma_{\omega 2} \quad (2.44)$$

Coefficient β in the transport equation for specific dissipation rate is calculated as equation 2.45.

$$\beta = F_1\beta_1 + (1 - F_1)\beta_2 \quad (2.45)$$

The model constant coefficients for the SST $k - \omega$ model are shown in Table 2.2 below. The SST $k - \omega$ model has two additional equations to solve compared to the $k - \varepsilon$ and $k - \omega$ models, making it slower but generally more accurate than the standard $k - \varepsilon$ and $k - \omega$ models.

Table 2.2: Table showing the constant values for the model coefficients in the SST $k - \omega$ turbulence model.

Coefficient	Value	Coefficient	Value
C_T	0.6	σ_{k1}	0.85
α_1	0.31	σ_{k2}	1
β^*	0.09	$\sigma_{\omega 1}$	0.5
β_1	0.075	$\sigma_{\omega 2}$	0.856
β_2	0.0828	κ	0.41

2.2.2 Lag EB k-epsilon

The Lag Elliptic Blending (EB) $k - \varepsilon$ model developed by Lardeau and Billard [13] is inspired by the concepts used in the Elliptic Blending-Reynolds-Stress Model (EB-RSM) developed by Manceau and Hanjalić [14], to create a robust model for complex flow configurations. The Lag EB $k - \varepsilon$ model solves four additional equations in comparison to the eight additional equations solved by the EB-RSM, making the Lag EB $k - \varepsilon$ model less computationally demanding. The Lag EB $k - \varepsilon$ model consists of three transport equations 2.46-2.48 and one equation for the blending factor, equation 2.49. The transport equation for the lag parameter φ , equation 2.48, and the equation for the blending factor, equation 2.49, are ideas taken from the EB-RSM and applied to the standard $k - \varepsilon$ model to get the Lag EB $k - \varepsilon$ model. These two additional equations increase the robustness and accuracy of the Lag EB $k - \varepsilon$ model for complex flows. The transport equation for φ also incorporates terms that model anisotropic turbulence, which contributes to the increased accuracy of the model.

The Lag EB $k - \varepsilon$ model has been shown to perform well in a multitude of flow configurations, both simple and complex [13]. Wiström and Wallace [15] have found that the Lag EB $k - \varepsilon$ model predicts convective heat transfer closer to a LES than the standard SST $k - \omega$ model in a backwards-facing step flow configuration. The transport equations for the Lag EB $k - \varepsilon$ turbulence model are shown in equations 2.46-2.48 and the function for the blending factor α is shown in equation 2.49. σ_k , σ_ε , σ_φ , $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$ are model coefficients and are found in Table 2.3.

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k \bar{v}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\frac{\mu}{2} + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho(\varepsilon - \varepsilon_0) \quad (2.46)$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial \rho \varepsilon \bar{v}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\frac{\mu}{2} + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{1}{T_e} C_{\varepsilon 1} P_\varepsilon - C_{\varepsilon 2} \rho \left(\frac{\varepsilon}{T_e} - \frac{\varepsilon_0}{T_0} \right) \quad (2.47)$$

$$\frac{\partial \rho \varphi}{\partial t} + \frac{\partial \rho \varphi \bar{v}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\frac{\mu}{2} + \frac{\mu_t}{\sigma_\varphi} \right) \frac{\partial \varphi}{\partial x_j} \right] + \rho \frac{\varepsilon_0 \varphi_0}{k_0} + P_\varphi \quad (2.48)$$

$$\frac{\partial}{\partial x_j} \left(L^2 \frac{\partial \alpha}{\partial x_j} \right) = \alpha - 1 \quad (2.49)$$

The turbulent dynamic viscosity is calculated as equation 2.50 where the turbulent time scale T is calculated as equation 2.51 using T_e as described in equation 2.52. The values for the model coefficients C_μ , C_T and C_t are found in Table 2.3.

$$\mu_t = \rho C_\mu \varphi k \min \left(T, \frac{C_T}{\sqrt{3} C_\mu \varphi \bar{s}} \right) \quad (2.50)$$

$$T = \sqrt{T_e^2 + C_t^2 \frac{\nu}{\varepsilon}} \quad (2.51)$$

$$T_e = \frac{k}{\varepsilon} \quad (2.52)$$

The turbulent length scale L used in the blending factor function (equation 2.49) is calculated as equation 2.53 and the specific timescale T_0 , used in the transport equation for the dissipation rate ε (Eq. 2.47), is calculated as equation 2.54. The terms k_0 , ε_0 and φ_0 are sustaining terms that counteract the decay of turbulence. The values for the model coefficients C_L , C_η and C_T are found in Table 2.3.

$$L = C_L \sqrt{\frac{k^3}{\varepsilon^2} + C_\eta^2 \sqrt{\frac{\nu^3}{\varepsilon}}} \quad (2.53)$$

$$T_0 = \max \left(\frac{k_0}{\varepsilon_0}, C_T \sqrt{\frac{\nu}{\varepsilon_0}} \right) \quad (2.54)$$

The equations for the production terms for k and ε are equations 2.55 and 2.56, where G_k and G_b are the terms for the turbulent production and buoyancy production, respectively. The turbulent production is calculated as 2.57 and the buoyancy production as 2.58. Υ_M is a compressibility modification, calculated as 2.59, to the production of k where c is the speed of sound and C_M is a model coefficient. The additional production term E is calculated according to equation 2.60. The model coefficients $C_{\varepsilon 1}$, $C_{\varepsilon 3}$, C_M and C_k are found in Table 2.3. In the equation for the additional production term (equation 2.60), the wall-normal direction is represented by n with indices j and k to clarify that it is the magnitude of $2s_{ij}$ that is computed. β is here the coefficient of thermal expansion, which is calculated according to equation 2.61 for ideal gases. Pr_t is the turbulent Prandtl number, g_i is the gravitational vector and $\bar{\theta}$ is the mean temperature.

$$P_k = G_k + G_b - \Upsilon_M \quad (2.55)$$

$$P_\varepsilon = G_k + C_{\varepsilon 3} G_b + \frac{1}{C_{\varepsilon 1}} E \quad (2.56)$$

$$G_k = \mu_t \bar{s}^2 - \frac{2}{3} \rho k \frac{\partial \bar{v}_j}{\partial x_j} - \frac{2}{3} \mu_t \left(\frac{\partial \bar{v}_j}{\partial x_j} \right)^2 \quad (2.57)$$

$$G_b = \beta \frac{\mu_t}{Pr_t} \left(\frac{\partial \bar{\theta}}{\partial x_j} g_i \right) \quad (2.58)$$

$$\Upsilon_M = \frac{\rho C_M k \varepsilon}{c^2} \quad (2.59)$$

$$E = C_k (1 - \alpha)^3 \nu \mu_t \frac{k}{\varepsilon} \left(\frac{\partial ||2s_{ij}n_j||n_k}{\partial x_k} \right)^2 \quad (2.60)$$

$$\beta = -\frac{1}{\rho} \frac{\partial \rho}{\partial \theta} \quad (2.61)$$

The production of the lag parameter φ (equation 2.62) utilizes the blending factor α to get the correct ratio of the functions f_w and f_h , found in equations 2.63 and 2.64 below. The Lag EB $k-\epsilon$ model accounts for the anisotropy in the turbulence through the introduction of the anisotropy tensor a_{ij} (equation 2.65) in the f_h function. The terms w_{ij} , $\tilde{W}_{ij}$, W_{ij}^s , $\hat{W}_{ij}$ and w_k^f are the vorticity tensor, absolute vorticity tensor, Spalart-Shur tensor, the modified absolute vorticity tensor and frame rotation rate tensor, respectively. E_{kij} is the three-dimensional permutation tensor. The values for the model coefficients $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, C_1^* , C_3^* , C_1 , C_4 and C_5 are found in Table 2.3.

$$P_\varphi = -(2 - C_{\varepsilon 1}) \frac{\varphi}{k} (G_k + G_b) + \rho (1 - \alpha^3) f_w + \rho \alpha^3 f_h \quad (2.62)$$

$$f_w = - \left(C_{\varepsilon 2} - 1 + 5 - \frac{1}{C_\mu} \right) \frac{\varphi}{T_e} \quad (2.63)$$

$$f_h = -\frac{1}{T_e} \left(C_1 + C_{\varepsilon 2} - 2 + C_1^* \frac{G_k + G_b}{\rho \varepsilon} \right) \varphi + \frac{C_{P3}}{T_e} + \frac{C_3^*}{\sqrt{2}} \varphi \bar{s} + \frac{1}{\bar{s}^2 T_e} \left[\frac{2}{C_\mu} (1 - C_4) a_{ij} s_{ij} - \frac{2}{C_\mu} (1 - C_5) a_{ij} \tilde{W}_{ij} \right] s_{ij} \quad (2.64)$$

$$a_{ij} = -2 \frac{\mu_t}{\rho k} \left[s_{ij} + \frac{2 - 2C_5}{C_1 + C_1^* + 1} \frac{2}{\sqrt{(s_{ij} + \hat{W}_{ij})(s_{ij} + \hat{W}_{ij})}} (s_{ij} \hat{W}_{ij} - \hat{W}_{ij} s_{ij}) \right] \quad (2.65)$$

$$\hat{W}_{ij} = \tilde{W}_{ij} - W_{ij}^s \quad (2.66)$$

$$\tilde{W}_{ij} = W_{ij} + E_{kij} w_k^f \quad (2.67)$$

$$W_{ij}^s = \frac{1}{\bar{s}^2} \left(s_{ij} \left(\frac{\partial s_{ij}}{\partial t} + \frac{\partial s_{ij}}{\partial x_j} \right) - \left(\frac{\partial s_{ij}}{\partial t} + \frac{\partial s_{ij}}{\partial x_j} \right) s_{ij} \right) \quad (2.68)$$

Kinematic blocking of the wall in the Lag EB $k - \varepsilon$ model is implemented through the coefficient C_{P3} and it utilizes a dampening function f_μ (equation 2.70) that is calculated using the timescale ratio η and the blending factor α . The timescale ratio is calculated according to equation 2.71 and the model coefficients C_μ and C_3 are found in Table 2.3.

$$C_{P3} = \frac{f_\mu}{C_\mu} \left(\frac{2}{3} - \frac{C_3}{2} \right) \quad (2.69)$$

$$f_\mu = \frac{\eta + \alpha^3}{\max(\eta, 1.87)} \quad (2.70)$$

$$\eta = \bar{s}T_e \quad (2.71)$$

Table 2.3: Model coefficients for the Lag EB $k - \varepsilon$ turbulence model. In $C_{\varepsilon 3}$, $v_\parallel$ and $v_\perp$ are the parallel and perpendicular velocity components to the gravitational vector g_i .

Coefficient	Value	Coefficient	Value
σ_ε	1.2	$C_{\varepsilon 2}$	1.9
σ_φ	1	$C_{\varepsilon 3}$	$\tanh\left(\frac{ v_\parallel }{ v_\perp }\right)$
σ_k	1	C_μ	0.22
C_1	1.7	C_η	75
C_3	0.8	C_k	2.3
C_4	0.625	C_L	0.164
C_5	0.2	C_M	2
C_1^*	0.9	C_T	1
C_3^*	0.65	C_t	4
$C_{\varepsilon 1}$	1.44		

2.3 Spatial discretization

Spatial discretization is the basis of the finite volume method (FVM). The fluid volume is discretized into a mesh of cells, which make up the fluid in the computational domain. The Navier-Stokes equations are solved iteratively for the mesh, where a value is stored in each cell, resulting in a mosaic-like solution of the entire domain. The size of an individual cell has a significant impact on the accuracy of the solution in the domain. With larger cells, the number of cells that fit in the domain is reduced, resulting in a coarser resolution of the domain. There are generally two categories of meshes: structured and unstructured. A structured mesh is ordered,

meaning that the cells are uniformly arranged. An unstructured mesh is built irregularly, containing cells of different shapes and sizes in the same region. One category of mesh can work better for a certain type of flow. An example of this is that a structured mesh might work better for a flow that follows the Cartesian directions closely than an unstructured mesh would, since the faces of each individual cell are oriented normal to the Cartesian directions.

A few common mesh types in CFD codes are tetrahedral, hexahedral, and polyhedral meshes. The tetrahedral mesh is built from pyramidal structures; they are cheap to generate and they are generally good at handling complex geometries. The big issue with tetrahedral meshes is that they need more cells, in comparison to the hexahedral and polyhedral meshes, to capture the geometry. The polyhedral mesh is great for complex geometries and is relatively fast to generate. The mesh consists of polyhedral cells where each cell has many cell faces and therefore many neighbors. This is a major benefit for accurately approximating the gradient in the flow and reducing the numerical diffusion of flows that are non-perpendicular to any of the cell faces [16]. The polyhedral mesh is converted from a tetrahedral mesh [7], inheriting the geometrical sensitivity of the tetrahedral mesh while reducing the number of cells in the mesh. In StarCCM+, the closest type of mesh to the hexahedral mesh is the trimmed cell mesh, which is made up of hexahedral cells in the core mesh. The hexahedral cells are trimmed to polyhedral cells near the geometric surface to capture the geometry. The bigger hexahedral cells are split into smaller cells when nearing the geometry surface, and if the geometry is simple (or consists of straight lines), the cells closest to the geometric surface will not be trimmed. Figure 2.1 gives a visual representation of how the spatial discretization of a square block is affected by the different meshing methods mentioned in this section. The meshes that are used in this work are the trimmed and polyhedral meshes, Figures 2.1b and 2.1c, generated using StarCCM+.

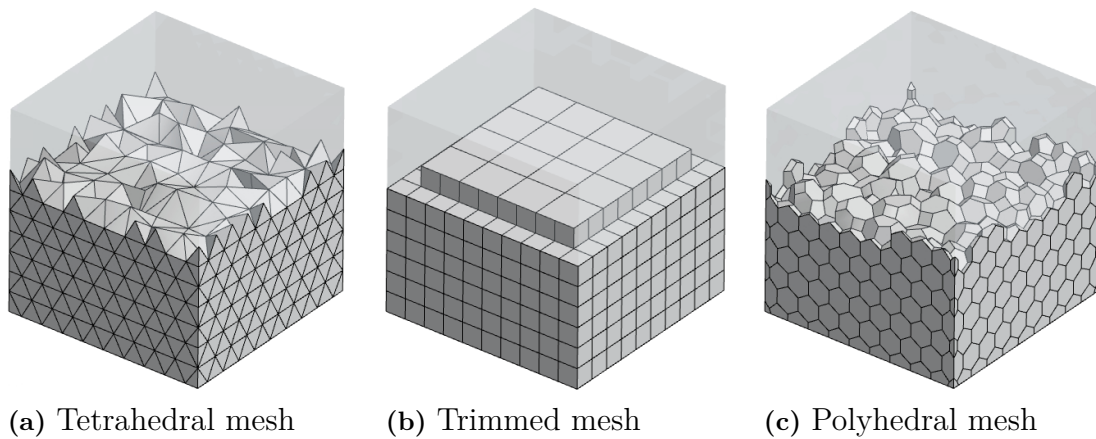


Figure 2.1: The three common meshing methods with the same settings used to discretize the same block.

2.3.1 Mesh convergence study through GCI

The Grid Convergence Index (GCI) is a measure used to determine the accuracy of the results for a given mesh and is based on the Richardson Extrapolation. It indicates how far off the solution is from its asymptotic value at a chosen point. The lower the GCI is, the closer to the asymptotic solution the simulation is for that point in the mesh [17]. This index is often used to estimate the magnitude of errors a coarser mesh can introduce and can help determine whether a finer mesh is worth the computational cost. The general approach for calculating the GCI is described in detail in [18] and the formulas used in this thesis are listed below.

The representative mesh size h is calculated from the volume of the fluid V_{fluid} and the total number of cells in the mesh N , according to equation 2.72.

$$h = \left(\frac{V_{fluid}}{N} \right)^{\frac{1}{3}} \quad (2.72)$$

The mesh refinement factor r between two meshes is calculated as a ratio between a coarser and a finer mesh, as shown in equation 2.73. According to Celik et al. [18], r should preferably be larger than 1.3.

$$r = \frac{h_{coarse}}{h_{fine}} \quad (2.73)$$

The order of convergence p is determined through iteration of equations 2.75-2.77, where ϵ is calculated according to equation 2.74 and ϕ_n indicates that the variable is taken from the n :th mesh. An initially guessed value is used as the first term in $q(p)$. The variable s is the sign of the ratio of the error ϵ between the meshes studied (see Eq. 2.77). Celik et al. [18] noted that a negative error ϵ is a sign of oscillatory convergence.

$$\epsilon_{21} = \phi_2 - \phi_1 \quad (2.74)$$

$$p = \frac{|ln|\frac{\epsilon_{32}}{\epsilon_{21}}| + q(p)|}{ln(r_{21})} \quad (2.75)$$

$$q(p) = ln\left(\frac{r_{21}^p - s}{r_{32}^p - s}\right) \quad (2.76)$$

$$s = 1 \cdot sign\left(\frac{\epsilon_{32}}{\epsilon_{21}}\right) \quad (2.77)$$

The extrapolated value ϕ_{ext} is calculated as in equation 2.78. The approximate relative error e_a is shown in equation 2.79 and the extrapolated relative error e_{ext} in equation 2.80. At last, the grid convergence index is calculated from the relative error and the mesh refinement factor as equation 2.81 shows. Note that the indexation is related to the mesh, so for instance: GCI^{21} is related to mesh 1 and 2.

$$\phi_{ext} = \frac{(r_{21}^p \phi_1 - \phi_2)}{r_{21}^p - 1} \quad (2.78)$$

$$e_a^{21} = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right| \quad (2.79)$$

$$e_{ext}^{21} = \left| \frac{\phi_{ext}^{12} - \phi_1}{\phi_{ext}^{12}} \right| \quad (2.80)$$

$$GCI^{21} = \frac{1.25e^{21}}{r_{21}^p - 1} \quad (2.81)$$

2.3.2 Cell size for highly resolved turbulence

A simplified figure showing an energy spectrum in logarithmic scale is shown in Figure 2.2 to give some visual context to this section. The properties of turbulent eddies depend on where they are located on the energy spectrum. Large eddies extract kinetic energy from the mean flow and have a big impact on the flow. They transfer the kinetic energy to smaller eddies in the inertial subrange. The eddies in the inertial subrange transfer the kinetic energy to the smallest eddies in the dissipation range of the energy spectrum, where the viscous stresses become too large and the energy is dissipated. This process, from the largest scales to the smallest Kolmogorov scales, is known as the “Cascade process”. This is a simplified description of the energy spectrum and cascade process; in reality, there are other contributing factors to the energy transport, and some energy is dissipated in all regions of the spectrum.

The turbulent length scale is a useful parameter for characterizing the size of a turbulent eddy and placing it in one of the ranges of the energy spectrum mentioned above [5]. For a well-resolved LES, an initial guess for a reasonable mesh can be made using the Taylor microscale and the Kolmogorov scale turbulence in the energy spectrum, see Figure 2.2. The mesh is found by limiting the maximum cell size as the Taylor microscales and the minimum cell size as the Kolmogorov scales [7]. The cell size of the polyhedral cells can be approximated as equation 2.82 according to the StarCCM+ user guide [7].

$$h_{cell} = 1.2 \cdot V_{cell}^{\frac{1}{3}} \quad (2.82)$$

There are several ideas on how the resolution of a LES can be measured. The most intuitive definition is to evaluate the solution using experimental results, which might not be feasible for rapid design since it requires real-world experimental data from the part that is being designed. If the energy spectrum of a fluid simulation exhibits a -5/3 decay (see Figure 2.2) it is an indication that the resolution is good enough to resolve a part of the turbulence in the inertial subrange. These methods have, however, been criticized by Davidson [19], whose work proved that those methods are flawed. Davidson suggests the use of two-point correlation to determine the mesh resolution. The two-point correlation for a coarse LES should span at least eight cells and a well-resolved LES should span a greater number of cells [19].

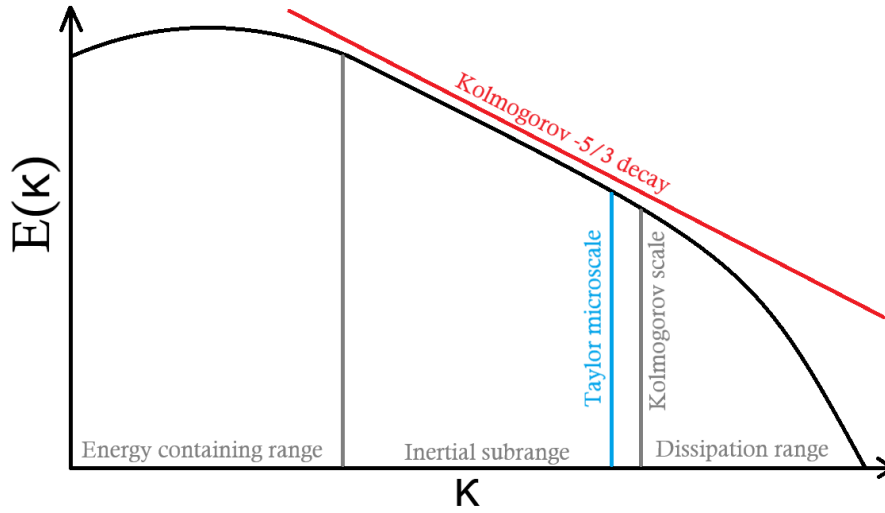


Figure 2.2: Energy spectrum with Taylor microscale and Kolmogorov scale marked. The Kolmogorov law $-5/3$ decay is also shown in this figure. Note that the Taylor microscale is placed in the inertial subrange.

2.3.3 Transition modelling

The transition region in the boundary layer is the region between the laminar and turbulent regions in the boundary layer, see Figure 2.3. A transition model combined with a turbulence model predicts the location of the transition region inside the boundary layer, determining where the switch from laminar to turbulent flow occurs as well as the size of the transition region in the simulation. The two transition models studied in this thesis are the γ transition model and the $\gamma - Re\theta$ transition model. Both of these models are based on correlations and are coupled with the SST $k - \omega$ turbulence model in StarCCM+ [7].

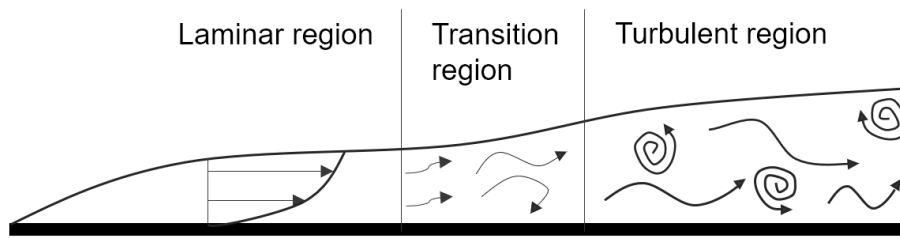


Figure 2.3: A representation of the boundary layer along a flat surface where the different regions are shown.

The $\gamma - Re\theta$ transition model solves two additional equations, the transport equation for intermittency γ (Eq. 2.83) and the transport equation for the transition momentum thickness Reynolds number $Re_{\theta t}$ (Eq. 2.84). Intermittency determines the time period for which the flow is turbulent, acting as the production term for turbulent kinetic energy. The transition momentum thickness Reynolds number is the criterion that determines the start of the transition and is based on the momentum thickness Reynolds number [20], the Reynolds number with characteristic

length set as the thickness of the boundary layer [21]. The $\gamma - Re\theta$ transition model is specifically formulated for unstructured meshes and [7]. In equation 2.83, $\sigma_f = 1$ is a model coefficient, P_γ is the production term, and E_γ is the destruction term. In equation 2.84, $\sigma_{\theta t} = 2$ is a model coefficient, $P_{\theta t}$ is the production term.

$$\frac{d\rho\gamma}{dt} + \frac{\partial\rho\gamma\bar{v}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_f} \right) \frac{\partial\gamma}{\partial x_j} \right] + P_\gamma - E_\gamma \quad (2.83)$$

$$\frac{d\rho\overline{Re_{\theta t}}}{dt} + \frac{\partial\rho\overline{Re_{\theta t}}\bar{v}_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\sigma_{\theta t}(\mu + \mu_t) \frac{\partial\overline{Re_{\theta t}}}{\partial x_j} \right] + P_{\theta t} \quad (2.84)$$

The γ transition model is a simplification of the $\gamma - Re\theta$ transition model. The only transport the γ transition model solves is the transport equation for the intermittency. This is the same equation that is solved in the $\gamma - Re\theta$ transition model (Eq. 2.83) [7].

2.4 Inviscid flux discretization schemes

Both of the inviscid flux discretization schemes investigated in this work, the Wess-Smith preconditioned Roe's scheme and the Liou's Advection Upstream Splitting Method + (AUSM+) scheme, are based on upwind differencing.

The Roe's discretization scheme is the standard scheme for the coupled solver in Star-CCM+. The scheme splits the flux at each face into parts containing characteristic information that propagates through the domain. The flux at each face f_f is calculated according to equation 2.85 below, where ΔQ_{ij} is a reconstructed solution vector from cells 0 and 1 in relation to either side of the face f . Matrix $|A|$ is defined by a diagonal matrix of eigenvalues and a modal matrix. This formulation of the Roe discretization scheme makes it second-order accurate [7].

$$f_{f,ij} = \frac{1}{2}(f_{0,ij} + f_{1,ij}) - \frac{1}{2}\Gamma|A_{ij}|\Delta Q_{ij} \quad (2.85)$$

Liou's AUSM+ flux-vector splitting inviscid flux discretization scheme applies the upwind concept to the convective part of the inviscid flux and the pressure part of the inviscid flux. The definition for the inviscid flux at each face is shown in equation 2.86. This discretization scheme emphasizes correct information propagation inside the fluid for convective processes since the calculation of fluxes is based on local flow characteristics [7].

$$f_{f,ij} = f_{ij}^{(c)} + p_{ij} \quad (2.86)$$

3

Method

3.1 Problem description

The geometry used in this thesis is a simplified representation of an internal bay in an aircraft, meaning that it does not exist in the physical world but is still a realistic representation. Therefore, there are no experimental data to be used as reference data for this geometry. A highly resolved SRH simulation was performed and used as the reference case in this work. Several simplified steady-state RANS simulations with different settings were performed to answer the research questions of the work through comparison between the RANS simulations and the reference case.

3.2 Geometry

A generalized part of internal airframe geometry, complete with internal structures and avionics devices, was provided by Saab. The idea is that the bottom plate and the top plate correspond to the bottom and the top skin plates of the airplane, respectively. The geometry of the fluid inside the bay (see Figure 3.1) was created by filling the inlet and outlet holes, creating a watertight compartment, and wrapping the internal geometry. The resulting geometry was then cleaned and repaired to prepare it for the simulations. The domain contains three avionics devices, three pipes, one inlet, one outlet and one leaking valve. The previously mentioned parts in the geometry are shown in Figure 3.1. The leakage is part of the geometry to simulate a worst-case scenario.

The flow from the leakage is of much higher temperature than the ambient fluid in the bay and it also has a velocity, making the flow mix faster with the ambient fluid. Simulating a worst-case scenario is useful for building robustness; in a case like this, the simulations could be used to show whether the ventilation is enough to keep the temperature inside the bay below the maximum temperatures for the avionics components. However, this is not studied in this thesis.

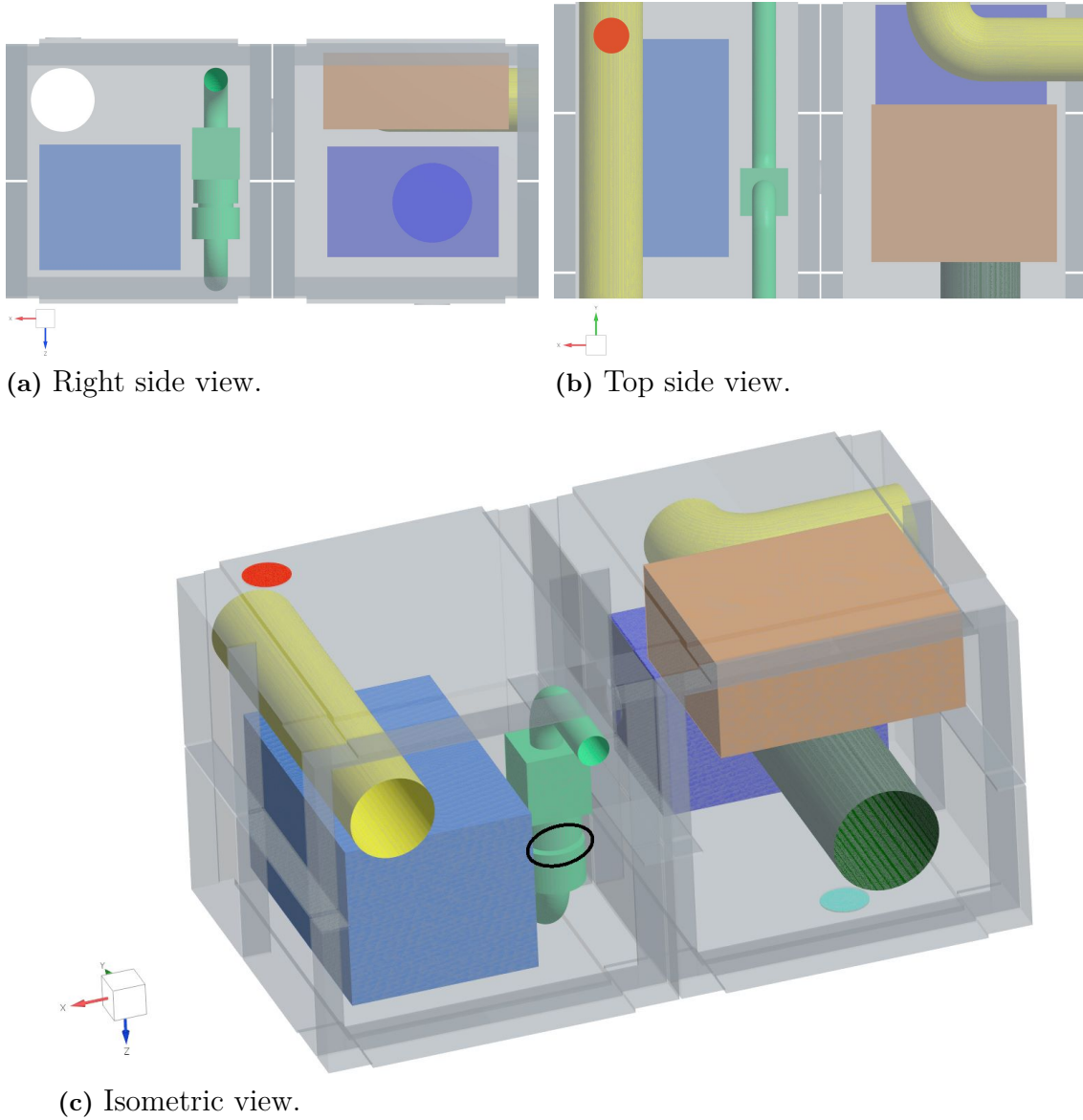


Figure 3.1: Three separate views of the fluid geometry with the objects in the domain colored. The aft wall is at the highest value on the x -axis and the forward wall is at the lowest value on the x -axis. The blue block in the aft bay represents avionics 1. The orange block in the forward bay represents avionics 2, the blue block in the forward bay represents avionics 3. The dark green pipe in the forward bay represents the avionics pipe. The yellow pipe in both the aft and forward bay represents an isolated pipe. The light green pipe in the aft bay represents a pipe with a leaking valve. The red circle on top of the aft bay and the cyan circle on the bottom of the forward bay are the Outlet and Inlet, respectively. The circled region on the pipe with the leaking valve in the aft bay in Figure 3.1c shows the location of the leaking valve.

3.3 Boundary conditions

The boundary conditions for the simulations are described in Table 3.1. The inlet is set at the bottom of the front bay and the outlet at the top of the rear bay, since the pressure on the bottom of the aircraft is greater than the pressure on the top when the aircraft is flying. All the remaining parts that are not mentioned in Table 3.1 were set as adiabatic non-slip walls. In Figure 3.1, the parts mentioned in Table 3.1 are colored and described. The inlet flow is set to follow the negative z axis and the leakage from the leaking valve flows along the positive z axis. The freestream air outside the bay follows the x direction in the figure. The airplane also follows the positive x axis, meaning that the greater x is, the further back the position is on the aircraft. The value for the thermal resistance R_k for the isolated pipe was found using a method from an internal study at Saab, where the heat transfer coefficient h_{htc} is set between the fluid inside the pipe and the pipe itself. The idea behind the method is to approximate the isolation without the need to model it as a solid to get the desired isolation effect.

Table 3.1: Boundary conditions for all the simulations in this work.

Boundary	Boundary Condition
Outlet	Pressure outlet: $p_{rel} = 0$ Pa, blocked reverse flow
Inlet	Massflow inlet: $\dot{m} = 0.012$ kg/s, Temperature: $\theta = 26.85$ °C
Leaking valve	Massflow inlet: $\dot{m} = 0.0016$ kg/s, $\theta = 350$ °C
Valve pipe	Non-slip wall, constant temperature: $\theta = 150$ °C
Isolated pipe	Non-slip wall, Convection Heat transfer coefficient: $h_{htc} = 1000$ W/m ² K Thermal resistance: $R_k \approx 0.1151$ m ² K/W
Avionics 1	Non-slip wall, Heat source: $P = 250$ W
Avionics 2	Non-slip wall, Heat source: $P = 250$ W
Avionics 3	Non-slip wall, Adiabatic
Avionics pipe	Non-slip wall, constant temperature: $\theta = 150$ °C

3.4 RANS simulations

3.4.1 Simulation setup

The general setups of the RANS simulations are presented in Table 3.2. The initial conditions for the RANS simulations are presented in Table 3.3. The fluid was treated as an ideal gas in all simulations. The configurations with the SST $k - \omega$ models have the quadratic constitutive relation enabled to account for anisotropic turbulence in the simulations.

Table 3.2: Table showing the general setup of each RANS simulation.

Simulation	Turbulence model	Mesh strategy	Prism layers
RANS _{V1}	SST $k - \omega$	Polyhedral	11
RANS _{V2}	Lag EB $k - \varepsilon$	Polyhedral	11
RANS _{V3}	SST $k - \omega$	Polyhedral	0
RANS _{V4}	Lag EB $k - \varepsilon$	Polyhedral	0
RANS _{V5}	SST $k - \omega$	Trimmed	11
RANS _{V6}	Lag EB $k - \varepsilon$	Trimmed	11
RANS _{V7}	SST $k - \omega + \gamma$ transition	Polyhedral	11
RANS _{V8}	SST $k - \omega + \gamma - Re\theta$ transition	Polyhedral	11
RANS _{V9}	SST $k - \omega$	Polyhedral	20
RANS _{V10}	RANS _{V8} with AUSM+ inviscid scheme	Polyhedral	11

Table 3.3: The initial conditions for the steady RANS simulations.

Pressure	0 Pa
Static Temperature	26.85 °C
Turbulence Intensity	0.01
Turbulent Velocity Scale	1 m/s
Turbulent Viscosity Ratio	10
Velocity	[0,0,0] m/s

The solver settings for the RANS simulations are shown in Table 3.4. The coupled solver was chosen due to its fast convergence. It is also proven to work better than the segregated solver for complex geometries where turbulence and heat transfer in the flows are important [22]. The second-order upwind convection scheme was chosen for the turbulence models. The second-order upwind discretization scheme was chosen to reduce the numerical diffusion in the simulation. The wall function used in the simulations was the all y^+ wall treatment for increased robustness since it blends the high y^+ and low y^+ wall functions. It emulates the high y^+ wall function for coarse mesh regions, the low y^+ wall function for fine mesh regions and blends the two functions between the coarse and fine mesh regions [7].

Table 3.4: The solver settings used for the RANS simulations in StarCCM+.

Solver settings	RANS simulations
Solver	Coupled solver
Spatial discretization scheme	2:nd order
Turbulence model convection scheme	2:nd order
Wall function	All y^+ wall treatment

3.4.2 Spatial discretization

Mesh convergence studies were performed on both the polyhedral mesh and the trimmed mesh to determine if the simulations are mesh independent or not. The mesh convergence study was done using three meshes of varying size for both the polyhedral mesh and the trimmed mesh. The sizes of the meshes used for the convergence study of the polyhedral mesh are presented in Table 3.5, where the variables related to the fine, medium and coarse meshes are indexed by the numbers 1, 2 and 3, respectively. The refinement factors between the meshes, presented in Table 3.6, are above 1.3, which was recommended by Celik et al. [18]. A convergence study was carried out on the trimmed mesh as well. The mesh sizes and refinement factors for the trimmed mesh are presented in Tables 3.7 & 3.8, respectively. The steady RANS simulations used for the mesh convergence study were run using the Lag EB $k - \varepsilon$ turbulence model.

Table 3.5: The mesh size and its representative mesh size used in the mesh convergence study of the polyhedral meshing method.

Mesh	Amount of cells	Representative mesh size
N_1	36.7 M	$h_1 = 0.00117$
N_2	8.8 M	$h_2 = 0.00188$
N_3	3.5 M	$h_3 = 0.00256$

Table 3.6: Mesh refinement factor and the related meshes for the polyhedral meshing method.

Mesher	Refinement factor
N_1 & N_2	$r_{21} = 1.608$
N_2 & N_3	$r_{32} = 1.363$

Table 3.7: The mesh size and its representative mesh size used in the mesh convergence study of the trimmed meshing method.

Mesh	Amount of cells	Representative mesh size
N_1	20.7 M	$h_1 = 0.00142$
N_2	6.4 M	$h_2 = 0.00209$
N_3	0.5 M	$h_3 = 0.00497$

Table 3.8: Mesh refinement factor and the related meshes for the trimmed meshing method.

Mesheres	Refinement factor
N_1 & N_2	$r_{21} = 1.477$
N_2 & N_3	$r_{32} = 2.372$

3.5 SRH simulation

The scale resolving hybrid model is used for the highly resolved turbulence simulation used as the reference case. The model can resolve turbulent scales down to the inertial subrange (see Figure 2.2) in the regions where LES is active. When the mesh resolution is too coarse to run LES, the SRH model switches to RANS and models the turbulence. The RANS model is mainly used close to the wall, as the mesh resolution requirements for LES increase drastically near the wall. This combination makes it possible to select which regions LES will be used in by creating regions in the domain where the mesh is refined to match the requirements for LES. The result of the model is a highly resolved flow for a lower resource cost than LES. This is the reason the SRH model was chosen as the reference case for this work.

3.5.1 Initial conditions

The initial conditions for the SRH simulation are set as the solution of the steady RANS simulation using the SST $k - \omega$ turbulence model and a polyhedral mesh with 11 prism layers. The fluid was treated as an ideal gas in the simulation.

3.5.2 Simulation setup

The SRH simulation was run using the SST $k - \omega$ turbulence model, with the quadratic constitutive relation to account for anisotropic turbulence. Since the turbulence model is widely used in industrial CFD applications, its performance in terms of strengths and weaknesses is well documented, and it has been proven to have good accuracy for its computational cost [9].

The solver settings for the SRH simulations are shown in Table 3.9. The second-order discretization scheme Hybrid-Bounded Central Differencing (Hybrid-BCD) was recommended by Siemens for LES simulations of complex turbulent flows since it adds a boundedness criterion that increases robustness of the simulation. The all y^+ wall function was recommended for the SRH simulation as well due to its robustness, as previously mentioned [7].

Table 3.9: The solver settings used for the SRH simulation in StarCCM+.

Solver settings	SRH simulation
Solver	Coupled solver
Spatial discretization scheme	2:nd order (Hybrid-BCD)
Temporal discretization	2:nd order
Turbulence model convection scheme	2:nd order
Wall function	All y^+ wall treatment

3.5.3 Spatial discretization

The cell size for the highly resolved turbulence case was determined through analysis of the Taylor microscales and the Kolmogorov scales of the initial steady-state RANS simulation. The size of the cells in the computational domain was made smaller than the local Taylor microscales to resolve the turbulence properly, but larger than the Kolmogorov scales, to reduce the computational cost of the simulation. This criterion was fulfilled by setting the average cell size in the domain to be larger than the Taylor microscales for the majority of cells, and creating local refinement regions where the cell size was above the Taylor microscales. Building the mesh in this manner optimizes the number of cells in the mesh, reducing both the required computational time and the required resources for the simulation.

The domain was set up to have 11 prism layers with a geometric stretching factor of 1.15. This choice was based on the recommendation by Ekman [23] to have at least 10 prism layers with no change in element size greater than 1.15 in the near-wall region, to accurately capture the boundary layer in a well-resolved LES simulation. These prism layer settings resulted in a wall Y^+ below 1 for the great majority of cells in the domain, as seen in Figure 3.2. The maximum wall Y^+ does not exceed 2.71, meaning that all the wall cells are located in the viscous sublayer [24] and that a low Y^+ wall function could be used. However, Star-CCM+ recommends using the all Y^+ wall function for the sake of robustness [7].



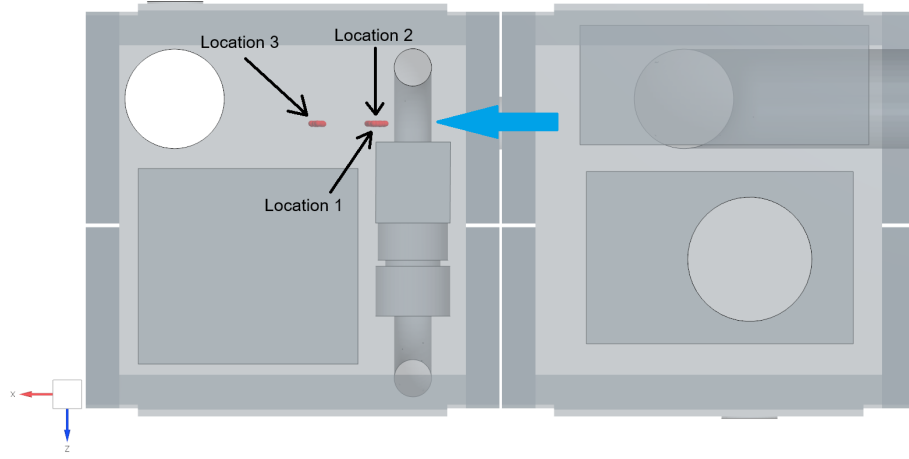
Figure 3.2: The wall Y^+ of the SRH simulation. The regions where the wall Y^+ are above one are located at impingement zones, the ventilation hole, or close to the inlet, leakage and outlet boundary conditions.

3. Method

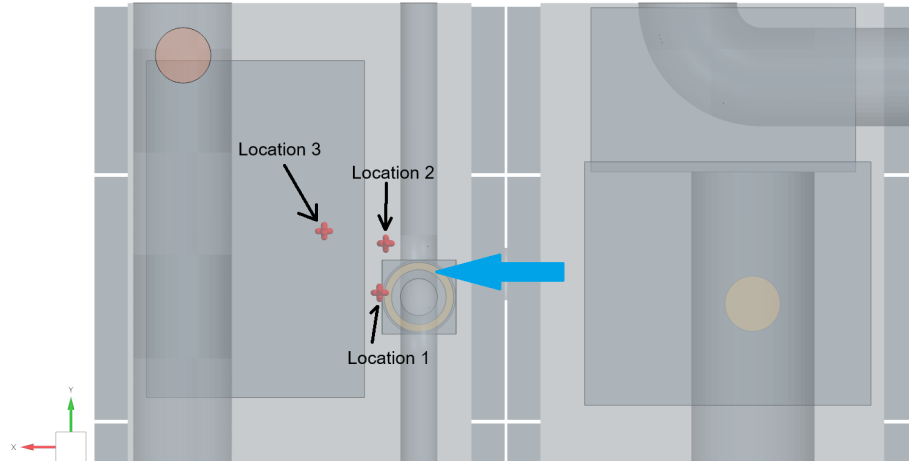
The regions where refinement was needed were found through the threshold function with the use of a custom field function, described in equation 3.1, where cells with values above 1 indicate areas where the cell size $V_{cell}^{\frac{1}{3}}$ is larger than the Taylor microscale λ . The meshing was carried out using the polyhedral meshing function in Star-CCM+ [7].

$$\frac{1.2 \cdot V_{cell}^{\frac{1}{3}}}{\lambda} \quad (3.1)$$

The resolution of the SRH simulation was determined through normalized two-point correlations of the fluctuating velocity components at three locations shown in Figure 3.3. Davidson [19] suggests that the resolution should be estimated from the two-point correlation in the poorest resolved direction.



(a) From the right side of the bay



(b) From the top side of the bay

Figure 3.3: The three locations used for evaluating the resolution of the simulation for SRH. The blue arrow represents the flow through the ventilation hole between the bays.

3.5.4 Temporal discretization

Initially, the solver timestep was intentionally set to be too large $\Delta t = 1 \cdot 10^{-4}$ s, resulting in a maximum CFL number of $CFL = 26$ in the domain. This timestep was used for the initial 2.5 seconds of the simulation, roughly 11 domain through flows. This timestep was used to let the large-scale turbulent structures develop and stabilize, and to remove the early transients caused by the switch from the steady RANS simulation to the SRH simulation. After these through flows, the timestep was reduced to $\Delta t = 6.666 \cdot 10^{-6}$ s and the maximum CFL number in the domain was reduced to below 4. The CFL number for the great majority of cells in the domain was below 1, where 0.0872% of the cells in the domain had a CFL above 1. This percentage was deemed acceptable since the majority of these cells with a CFL above one were located at impingement zones or close to the inlet and outlet boundaries, see Figure 3.4. Through monitoring of the residuals and instantaneous velocity, the required number of inner iterations for the SRH simulation was determined to be 5, as the residuals and velocity had stabilized, as is seen in Figure 3.5. The SRH simulation with the smaller timestep was run for 1.2 seconds, roughly 5 through flows of the domain, to let the small-scale turbulence develop.

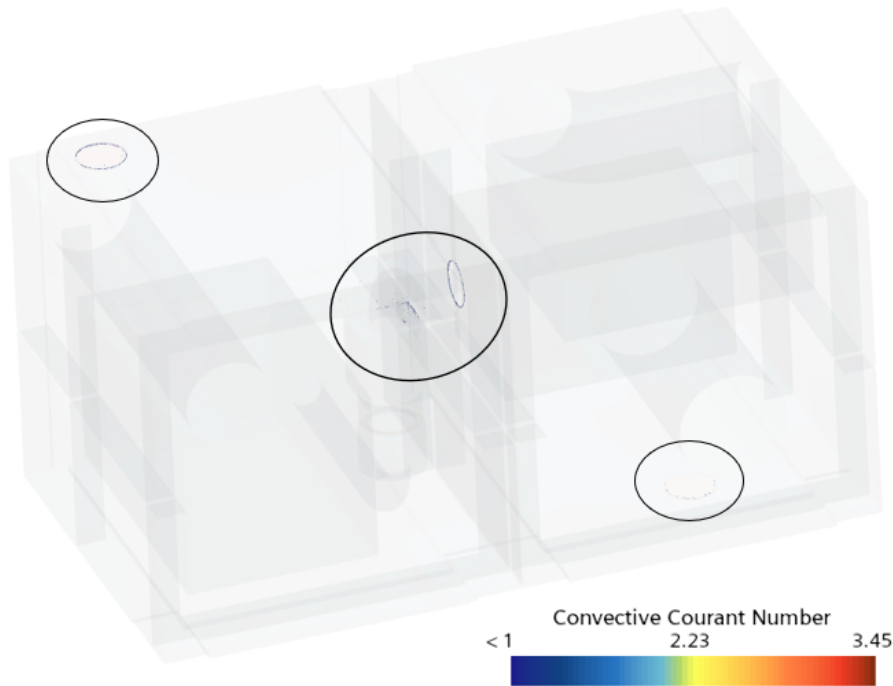
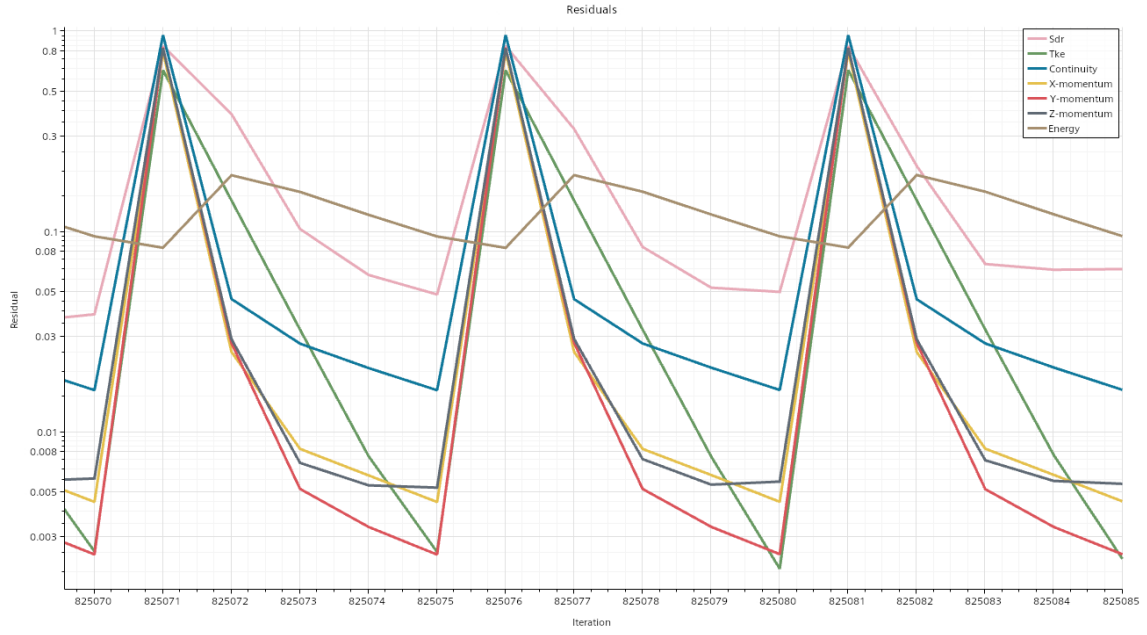
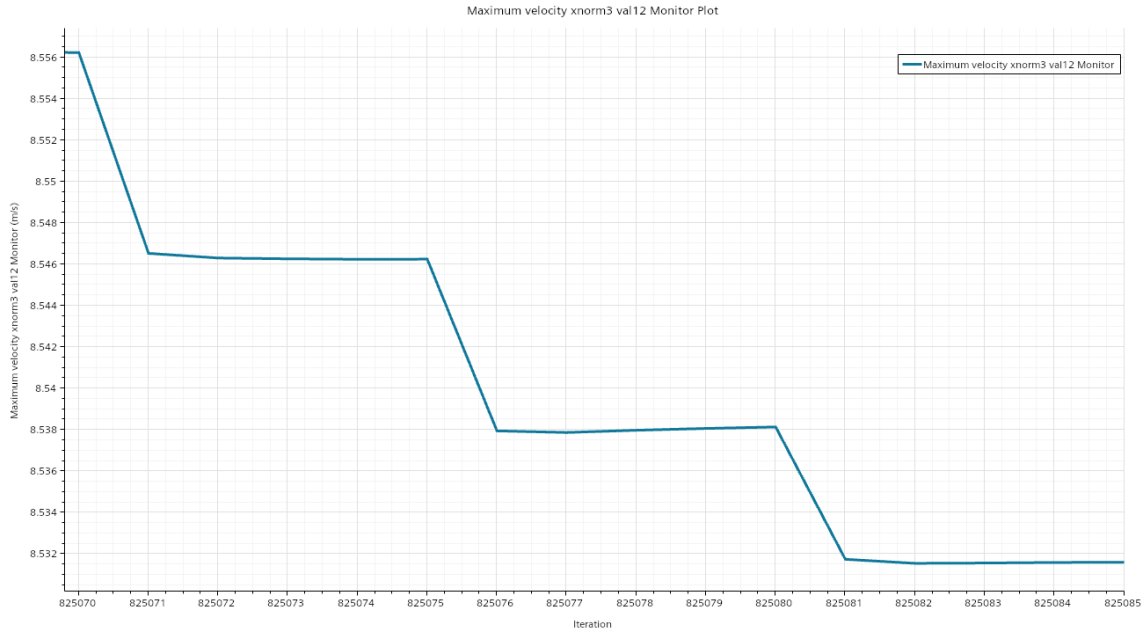


Figure 3.4: Figure showing the Convective Courant number (CFL) in the domain. The regions where cells with a CFL number larger than 1 are circled in this figure.

3. Method



(a) Residuals of the SRH simulation. As can be seen, the majority of the residuals drop more than one order of magnitude, with the exception of turbulent kinetic energy, which drops by barely half an order of magnitude. The turbulent kinetic energy graph did not drop more when more inner iterations were used, so it was determined to have stabilized after 5 inner iterations.



(b) Instantaneous velocity of a point in the SRH simulation. As is seen in the figure, the drop for each new timestep is quite large in comparison to the drop from the inner iterations. The velocity barely changes for the five inner iterations, meaning that the solution for each timestep is stable.

Figure 3.5: Plots showing the behaviour of the residuals for the SRH simulation as well as the behaviour of the velocity in a point in the domain.

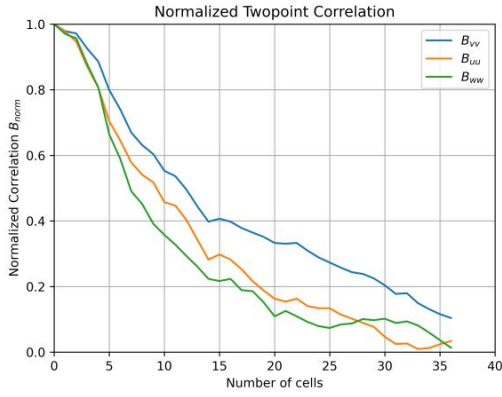
4

Results and Discussion

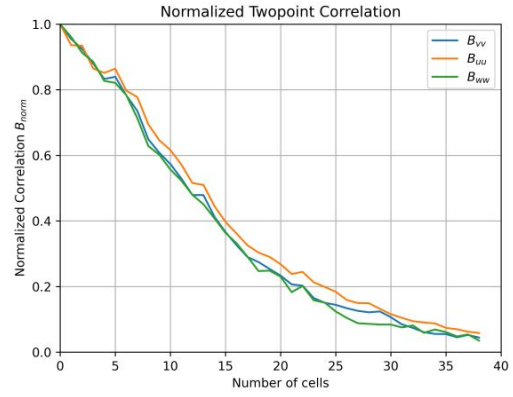
4.1 SRH mesh resolution

Figures 4.1-4.3 below show the two-point correlations in the x and y directions at three different locations (see Figure 3.3) in the domain where turbulence is present. Davidson [25] used 0.1 as the criterion for decorrelation, suggesting that a well-resolved simulation should be correlated for at least 8 cells, preferably more. Davidson [19] argues that a two-point correlation that drops to 0.4 within 4 cells implies poor resolution.

Using the criterion of 0.1 and keeping in mind the second argument, the resolution in location 1 can be considered sufficient, as the number of correlated cells well exceeds 8 cells in both the x and y directions. In location 2, the turbulence in the x direction seems to be well resolved. The two-point correlation in the y direction drops to 0.4 in only 4 cells, suggesting that the resolution is coarse but still resolved in this region. In location 3, the two-point correlation in the x direction suggests well-resolved turbulence. The correlation in the y direction at location 3 is slightly better than that in location 2, but the resolution is still quite coarse, with the two-point correlation dropping to 0.4 over just 5 cells. It is, however, correlated for more than 8 cells, suggesting sufficient resolution. To summarize, the two-point correlations suggest that the mesh is sufficient but could be further refined. The resolution was deemed sufficient for using the SRH simulation as the reference case in this work.

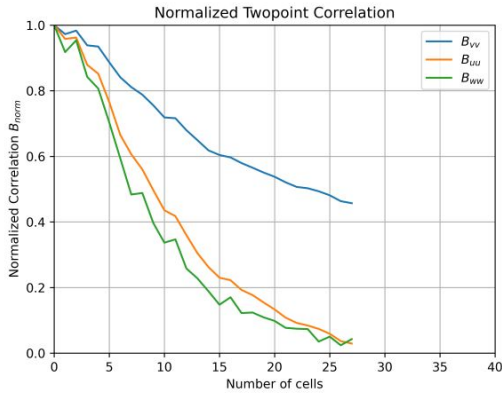


(a) Two-point correlation from velocities in x direction.

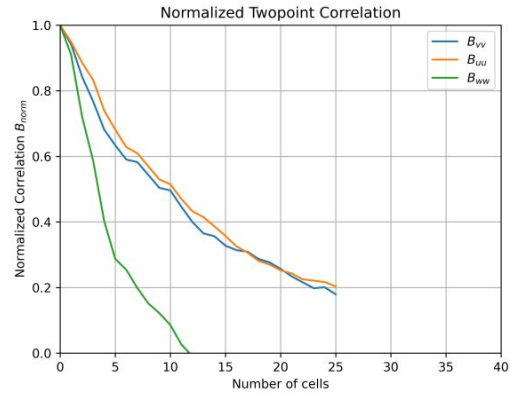


(b) Two-point correlation from velocities in y direction.

Figure 4.1: Two-point correlations of the fluctuating velocity components in turbulent structures residing at location 1 in the domain.

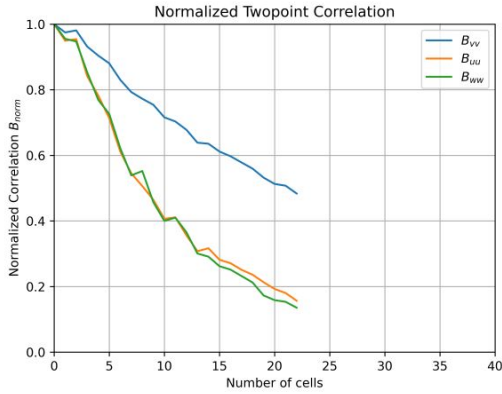


(a) Two-point correlation from velocities in x direction.

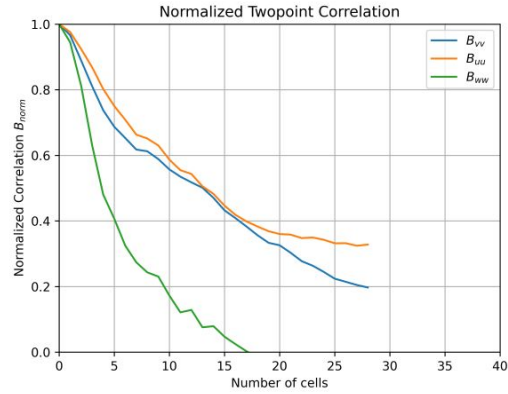


(b) Two-point correlation from velocities in y direction.

Figure 4.2: Two-point correlations of the fluctuating velocity components in turbulent structures residing at location 2 in the domain.



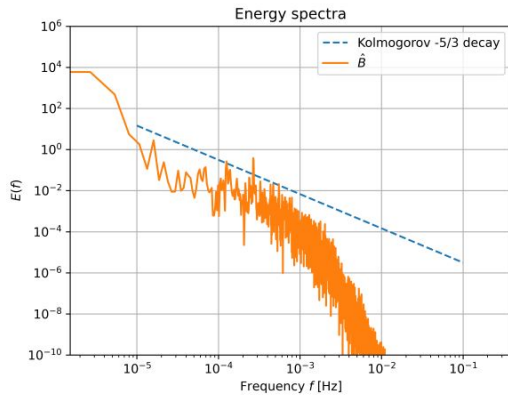
(a) Two-point correlation from velocities in x direction.



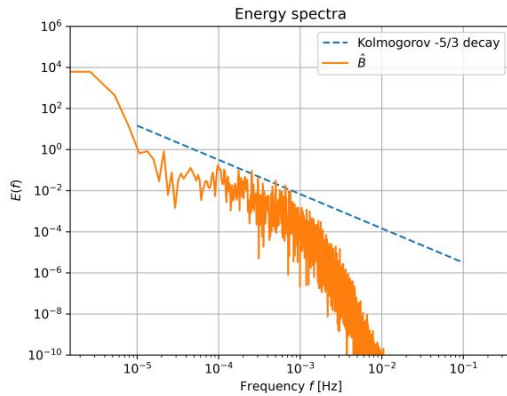
(b) Two-point correlation from velocities in y direction.

Figure 4.3: Two-point correlations of the fluctuating velocity components in turbulent structures residing at location 3 in the domain.

Figures 4.4-4.6 show the plotted energy spectra calculated from the Fast Fourier Transform (FFT) of the autocorrelation of the velocities at the three locations, together with the $-5/3$ decay of the Kolmogorov law. From the aforementioned figures, it is seen that the turbulence in the energy spectra exhibits a $-5/3$ decay over almost two orders of magnitude on the frequency axis at all three locations, suggesting that LES resolves a large part of the inertial subrange in all three locations. This implies that the simulation resolves a part of the turbulence in the inertial subrange at these locations, i.e. the resolution is sufficient [25].



(a) Energy spectra from autocorrelation of velocities in x direction



(b) Energy spectra from autocorrelation of velocities in y direction

Figure 4.4: Energy spectra based on the autocorrelation of the fluctuating velocity at location 1. The $-5/3$ decay based on the Kolmogorov law is plotted as the dashed line.

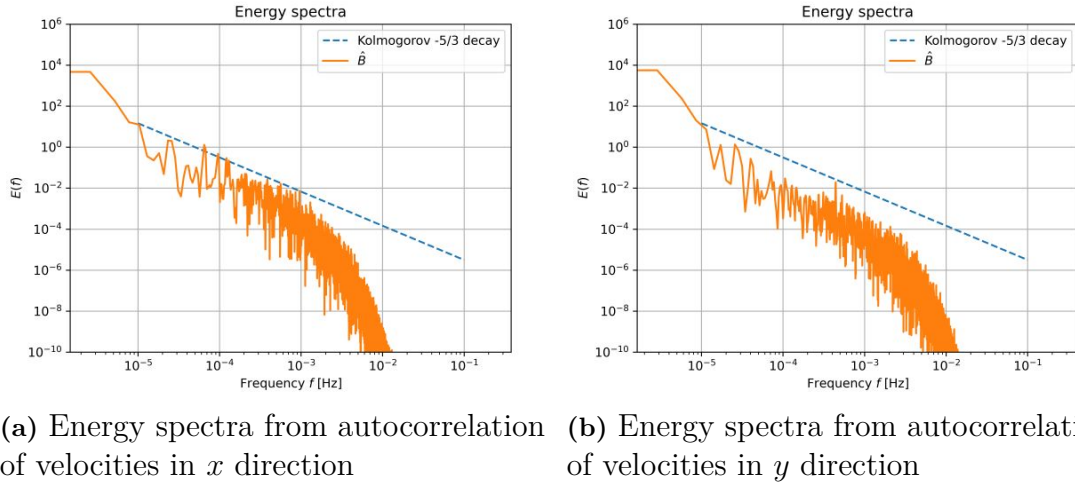


Figure 4.5: Energy spectra based on the autocorrelation of the fluctuating velocity at location 2. The $-5/3$ decay based on the Kolmogorov law is plotted as the dashed line.

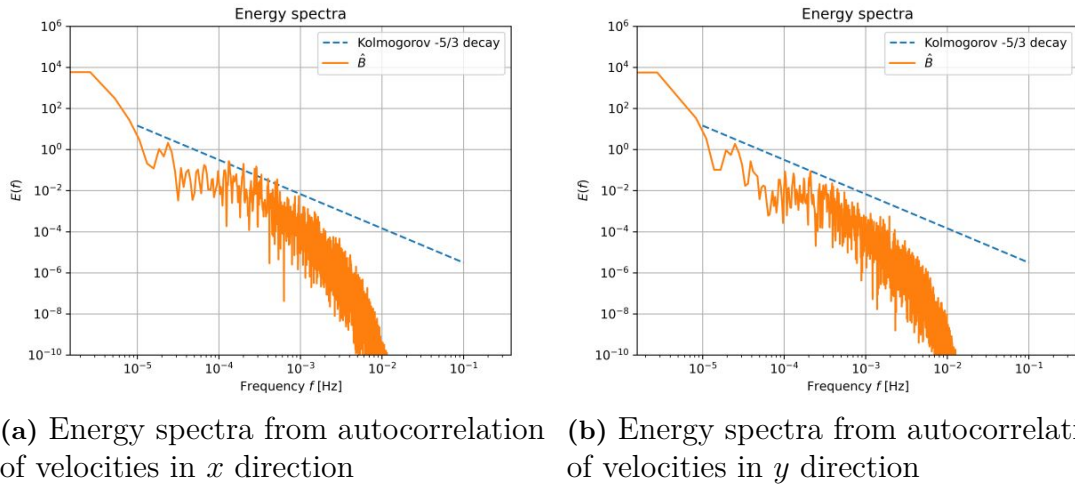


Figure 4.6: Energy spectra based on the autocorrelation of the fluctuating velocity at location 3. The $-5/3$ decay based on the Kolmogorov law is plotted as the dashed line.

The behaviour of the shielding function f_s was studied to ensure that LES was active in the turbulent regions in the domain. A cross-sectional view of the shielding function is shown in Figure 4.7. The shielding function approaches zero close to the walls, forcing the SRH model to use the RANS model at the wall as expected. The shielding function is 1 for the majority of the wake from the ventilation hole flow, meaning that the wake is almost completely simulated using LES. In the regions where the shielding function is between 0 and 1, the SRH model uses a blend of RANS and LES to solve the flow. The shielding function forces a large part of the domain to be solved in RANS mode. For future simulations, it might be worth

tuning the shielding function to switch to RANS mode later, which might reduce the robustness of the SRH model, or to refine the mesh even further. The two locations near the bottom wall of the avionics pipe in the forward bay, where the shielding function is 0, suggest that further mesh refinement is needed for LES mode in these locations.

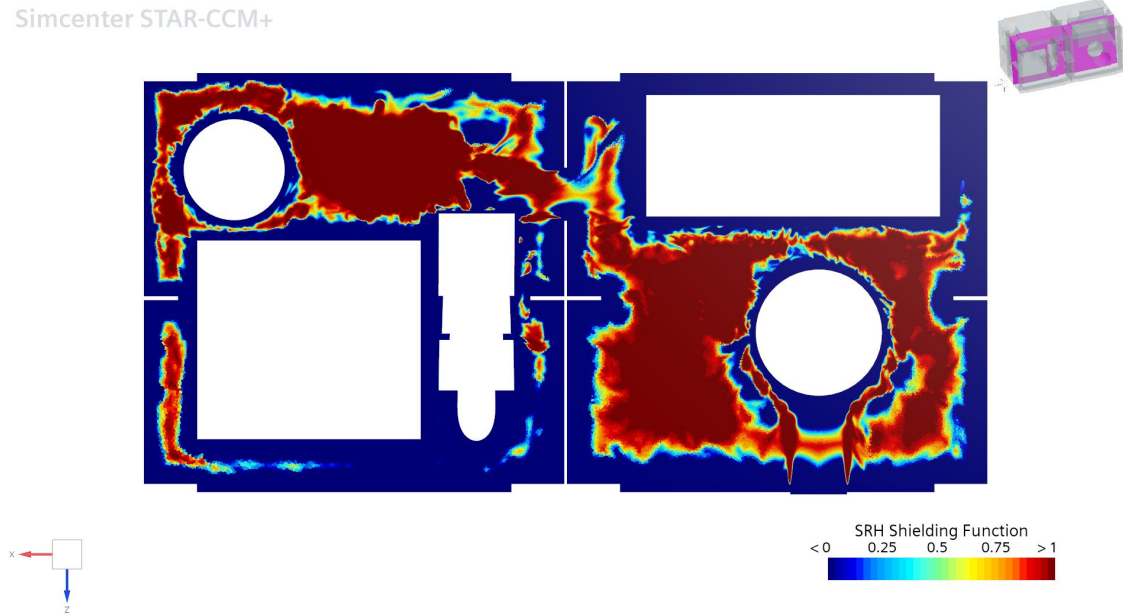


Figure 4.7: A cross-sectional view of the shielding function to visualize where LES is active and where RANS is active. Pure LES mode is activated when the shielding function is 1. Pure RANS mode is activated when the shielding function is 0. When the shielding function is between 0 and 1, the SRH simulation blends the two simulation modes. The location of the cross-section is shown in the top right corner of the figure.

4.2 RANS mesh convergence study

The mesh convergence study through GCI of the polyhedral and trimmed meshes with prism layers in Tables 4.1 and 4.2, respectively, shows that the meshes converge slowly for the heat transfer value of the avionics pipe and the isolated pipe. The error variables ε_{21} and ε_{32} are negative in both the polyhedral and the trimmed mesh studies. This is a sign of oscillatory convergence according to Celik et al. [18]. In general, the differences in GCI between the three polyhedral meshes and the three trimmed meshes were minimal, except for the heat transfer of the avionics pipe in the trimmed mesh. The chosen mesh sizes for the polyhedral mesh and the trimmed mesh were the medium meshes, containing 8.8 million cells and 6.4 million cells, respectively.

Table 4.1: Results from GCI analysis of surface-averaged heat transfer and mean temperature for three different polyhedral meshes of the domain. The variables $\bar{\theta}_{outlet}$ and $\bar{\theta}_{fluid}$ denote the surface-averaged mean temperature of the outlet and volume-averaged mean temperature of the fluid, respectively.

	$HT_{AvionicsPipe}$	$HT_{IsolatedPipe}$	$\bar{\theta}_{outlet}$	$\bar{\theta}_{fluid}$
ϕ_1	−300.71 W	−308.74 W	147.2 °C	112.5 °C
ϕ_2	−301.76 W	−307.78 W	146.4 °C	110.7 °C
ϕ_3	−296.87 W	−308.14 W	145.9 °C	108.6 °C
p	4.629	0.810	0.385	1.540
ϵ_{32}	4.89 W	−0.34 W	−0.55 °C	−2.10 °C
ϵ_{21}	−1.05 W	0.54 W	−0.73 °C	−1.79 °C
ϕ_{ext}^{21}	−300.58 W	−309.51 W	150.8 °C	114.1 °C
e_a^{21}	0.35 %	0.18 %	0.49 %	1.59 %
e_{ext}^{21}	0.04 %	0.38 %	2.41 %	1.45 %
GCI^{32}	0.64 %	0.49 %	3.72 %	3.89 %
GCI^{21}	0.05 %	0.47 %	3.08 %	1.84 %

Table 4.2: Results from GCI analysis of surface-averaged heat transfer and mean temperature for three different trimmed meshes of the domain.

	$HT_{AvionicsPipe}$	$HT_{IsolatedPipe}$	$\bar{\theta}_{outlet}$	$\bar{\theta}_{fluid}$
ϕ_1	−304.73 W	−305.87 W	147.3 °C	113.0 °C
ϕ_2	−291.91 W	−309.30 W	146.7 °C	110.2 °C
ϕ_3	−233.83 W	−295.75 W	139.6 °C	101.6 °C
p	1.097	0.157	2.419	0.530
ϵ_{32}	58.09 W	13.55 W	−7.1 °C	−8.6 °C
ϵ_{21}	12.82 W	−3.44 W	−0.6 °C	−2.8 °C
ϕ_{ext}^{21}	−328.73 W	−402.32 W	147.7 °C	125.1 °C
e_a^{21}	4.21 %	1.12 %	0.41 %	2.45 %
e_{ext}^{21}	7.30 %	21.54 %	0.26 %	9.65 %
GCI^{32}	15.76 %	37.59 %	0.85 %	16.84 %
GCI^{21}	9.84 %	22.15 %	0.33 %	13.36 %

The convergence of the outlet temperature was studied between the three mesh sizes for both the polyhedral and trimmed meshes, as shown in Figure 4.8. The plots in the figure are polygonal regressions of the three points as the values oscillate between meshes, as is seen in the GCI studies above. The temperature differs minimally between the coarsest and finest of the polyhedral meshes, suggesting that even the coarsest mesh at 3.5 million cells would be sufficient. The trimmed meshes show a larger difference in temperature, which is most likely because the coarsest mesh only contains 0.5 million cells. Still, the temperature only differs by around 7 °C between the coarsest and finest of the trimmed meshes, suggesting that the coarsest mesh could be sufficient with such a small error. Note that the medium meshes for both the polyhedral mesh and the trimmed mesh show a similar temperature, even though the trimmed mesh is coarser than the polyhedral mesh.

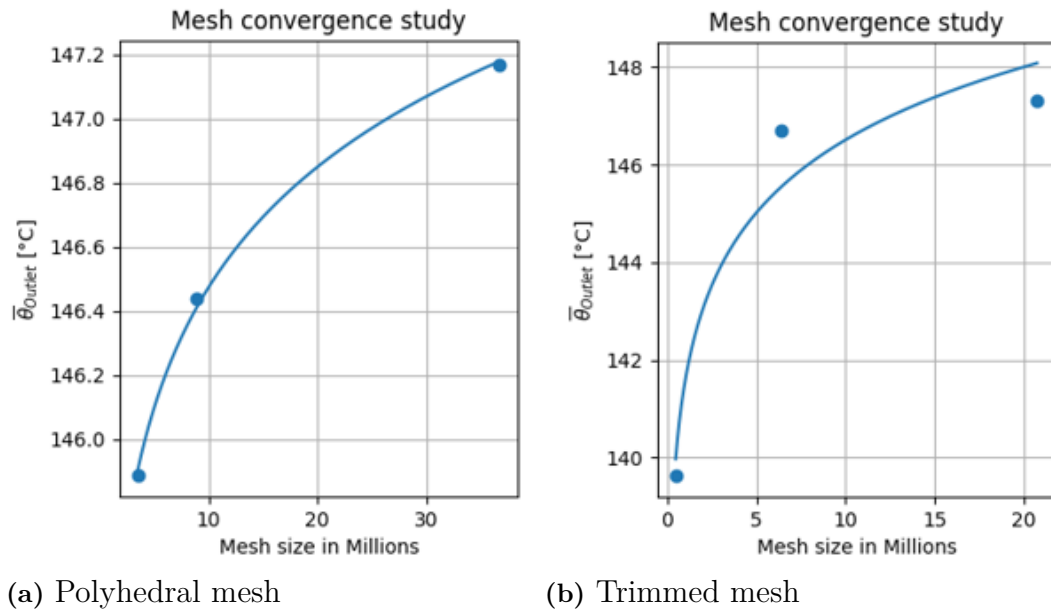


Figure 4.8: The convergence of the surface-averaged temperature at the outlet for the different mesh sizes used for the polyhedral and trimmed cell meshes.

4.3 Simulation results

4.3.1 SRH results

Figure 4.9 shows a cross-section of the relative pressure in the internal bay. The forward bay (right side in the figure) has a higher pressure than the aft bay (left side in the figure). The pressure difference is driving the flow through the ventilation hole from the forward bay to the outlet in the aft bay. A highly turbulent wake is evident in the flow leaving the ventilation hole.

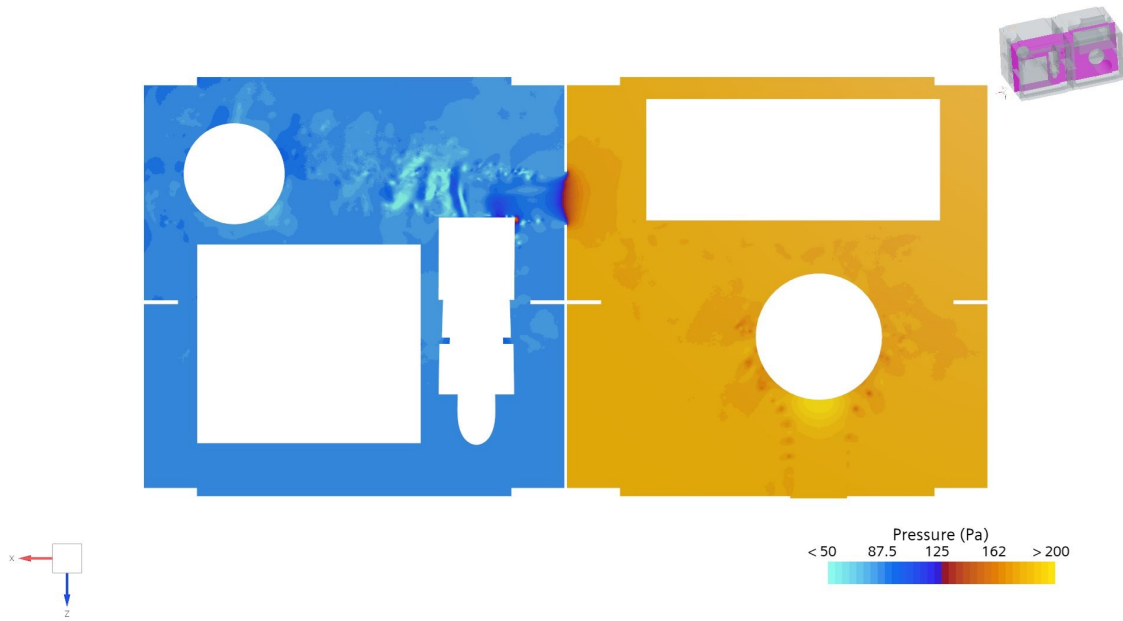


Figure 4.9: A cross-sectional view of the instantaneous relative pressure field in the bay.

The instantaneous velocity and temperature fields of the SRH simulation are time-averaged in order to compare the fields of the SRH simulation to those of the RANS simulations. The fields are shown in Figures 4.10 and 4.11. Three wake regions can be seen in the instantaneous velocity field in Figure 4.10. The first wake region is in the forward bay after the avionics pipe. The second one is located after the ventilation hole. The last wake region is created when the wake region from the ventilation hole hits the isolated pipe in the aft bay, creating a third wake region. In the figure showing the time-averaged velocity field (Figure 4.11), the wake regions are steady, clearly showing the main path of the flow.

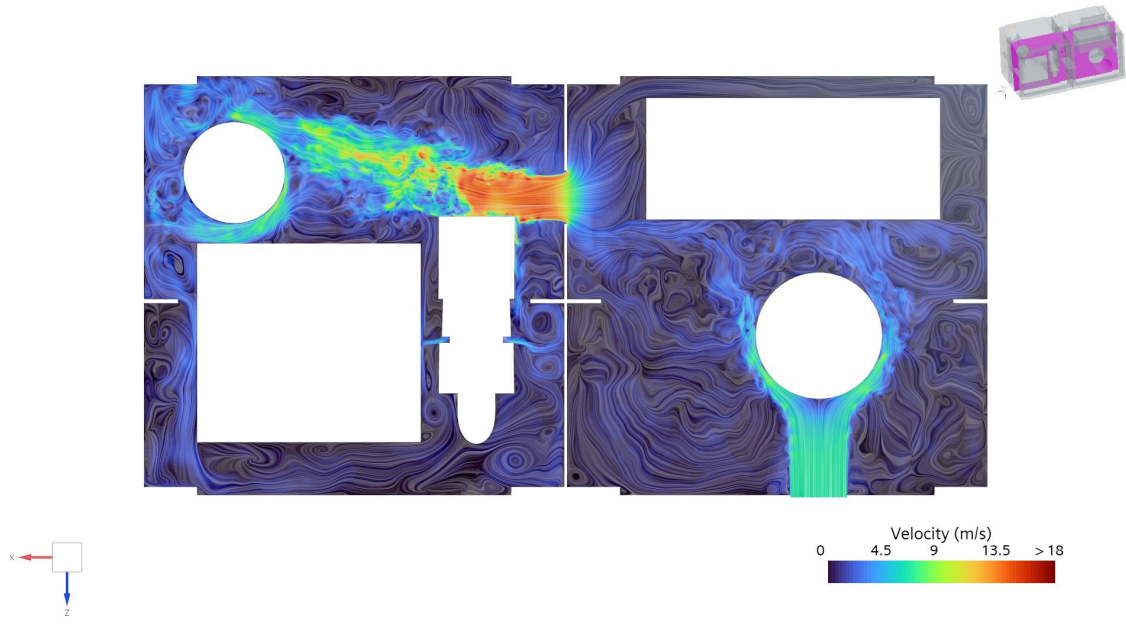


Figure 4.10: A cross-sectional view of the instantaneous velocity field of the SRH simulation.

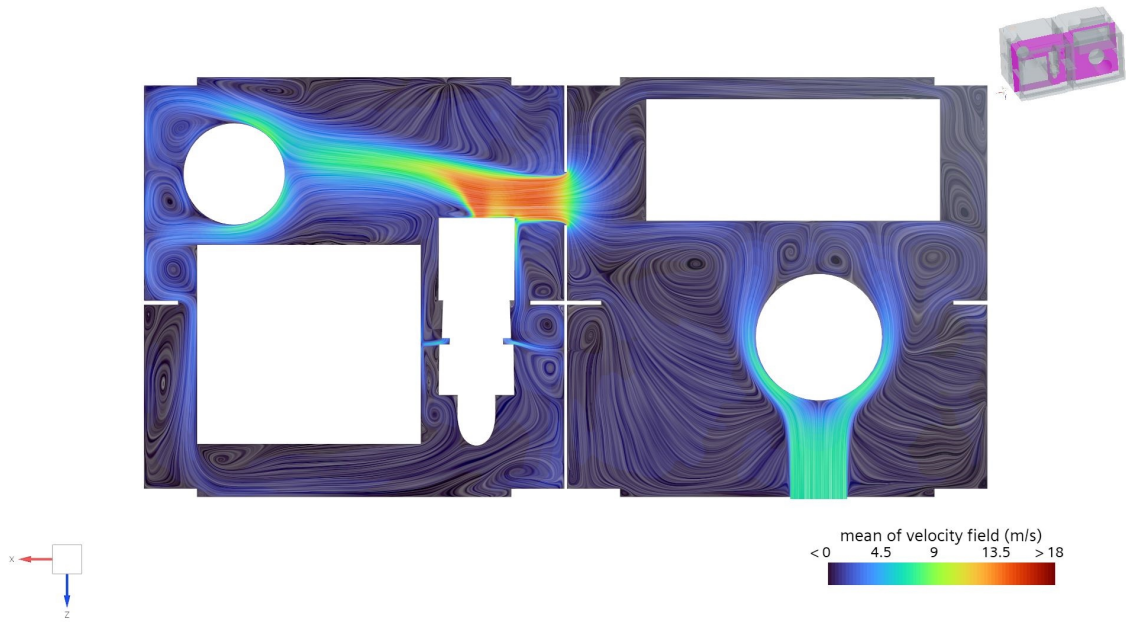


Figure 4.11: A cross-sectional view of the time-averaged velocity field of the SRH simulation.

In the instantaneous temperature field, Figure 4.12, a slower-moving turbulent flow structure can be observed above avionics 2, which is located at the top of the forward bay. This structure moves towards the ventilation hole and creates a separation bubble at the upper aft corner of avionics 2. The slower-moving flow structure can be seen in the time-averaged temperature field in Figure 4.13 as a steady temperature gradient on the top side of avionics 2. The separation bubble in the aft of avionics 2 in the instantaneous temperature field is clearly visible in the time-averaged temperature field, suggesting that the bubble resides in the domain for some time. The cold jet and wake of the jet from the forward bay to the aft bay is seen hitting the isolated pipe and spreading along the aft wall of the internal bay, cooling the isolated pipe and avionics 1. The average temperature in the aft bay is much higher than the average temperature in the forward bay. This is most likely due to the leakage from the valve pipe in the aft bay, where the leaking high-temperature airflow is spreading the heat through mixing in the fluid. The higher temperature is expected since the cooling flow from the ventilation hole is already heated from the forward bay and thus does not have the capacity to cool down the aft bay. A slower-moving hot fluid structure at the bottom of avionics 1, moving from the aft to the forward, is seen in Figure 4.12. The phenomenon results in a line of higher temperature at the bottom of the avionics in the time-averaged temperature field in Figure 4.13.

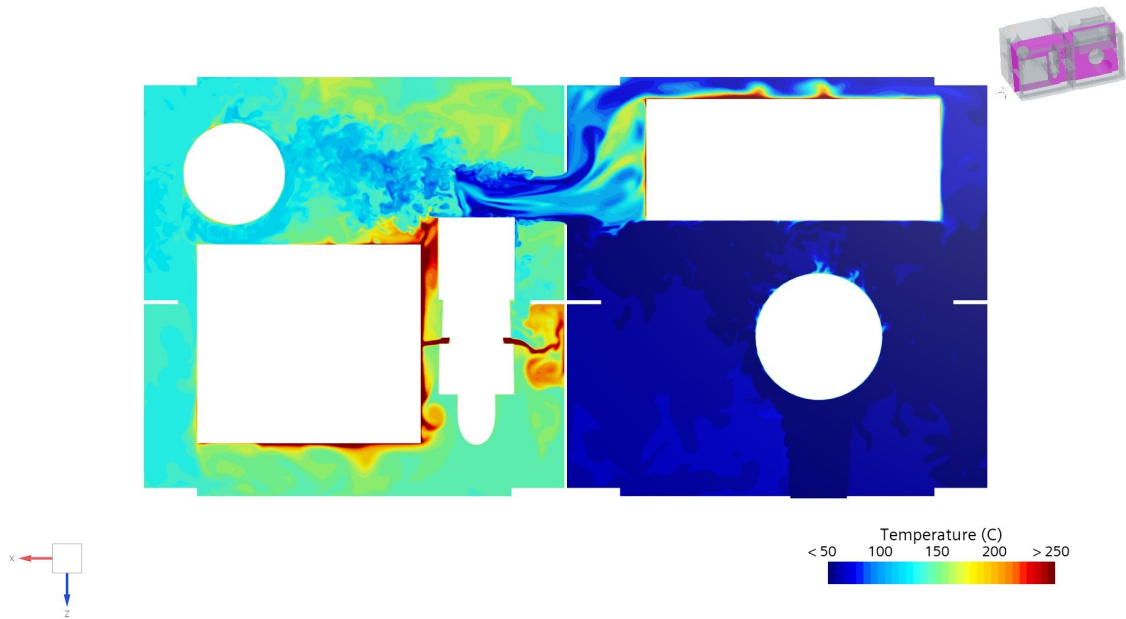


Figure 4.12: A cross-sectional view of the instantaneous temperature of the SRH simulation.

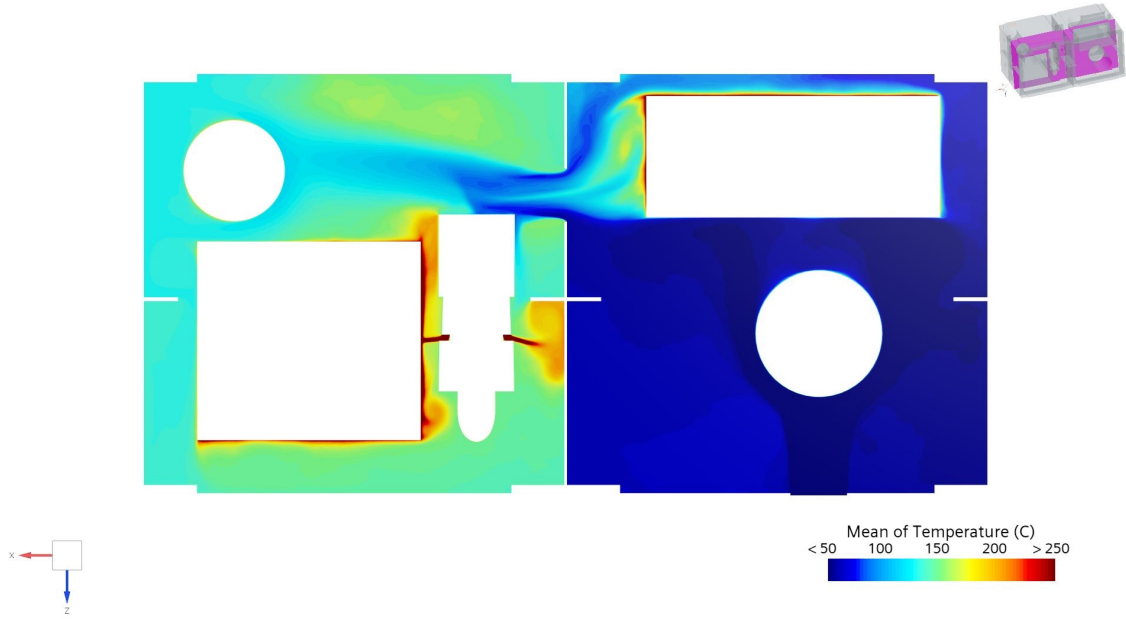


Figure 4.13: A cross-sectional view of the time-averaged temperature of the SRH simulation.

4.3.2 Numerical comparison of the RANS simulations

Table 4.3 shows the surface-averaged mean temperature of the avionics and the outlet. In this case, the mean temperature refers to the average temperature over 1000 iterations after the simulations had stabilized. Averaging was necessary since the temperature oscillated slightly around a value between iterations.

The temperature of avionics 1 is best predicted by the RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and a polyhedral mesh with 11 prism layers, being 4.1 °C off from the reference. For avionics 2, the temperature is best predicted by the RANS simulation using the SST $k - \omega$ model without prism layers, being only 0.2 °C off compared to the reference. The most accurate simulation for predicting the temperature of avionics 3 is the simulation using the Lag EB $k - \varepsilon$ turbulence model and a polyhedral mesh with prism layers. The isolated pipe temperature was best predicted by the SST $k - \omega$ model on the polyhedral mesh with prism layers. The outlet temperature was best predicted by two SST $k - \omega$ models, the one without a transition model and the one with the γ transition model.

The total error values presented in Table 4.3 were found by summing the absolute values of the errors for each simulation. In general, the simulation with the smallest total error in surface-averaged temperature presented in Table 4.3 is the RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model, a polyhedral mesh with 11 prism layers, and Liou's AUSM+ inviscid flux discretization scheme. The simulation with the SST $k - \omega$ turbulence model on the polyhedral mesh with 20 prism layers showed a slight reduction in total error by 1 °C. There is some doubt about how well this result would compare to actual experimental data instead of the reference simulation, since the reference simulation in this work uses RANS with 11 prism layers close to the walls, which most likely skews the result.

Surprisingly, the simulation using the SST $k - \omega$ turbulence model on a polyhedral mesh without prism layers produced a similar result to that of the simulation using the SST $k - \omega$ turbulence model on a trimmed mesh with 11 prism layers when studying the total error. This result suggests that the SST $k - \omega$ turbulence model works much better on a polyhedral mesh than on a trimmed mesh. Another interesting observation is that the total error for the simulations using the Lag EB $k - \varepsilon$ model on the polyhedral mesh with and without prism layers only differs by 6.1 °C, which is still a quite small difference. In other words, the Lag EB $k - \varepsilon$ turbulence model does not seem to be that affected by prism layers.

Table 4.3: Surface-averaged mean temperature of the parts inside the bay and of the outlet for the SRH simulation. The RANS simulation values are the error compared to the reference SRH simulation, where a negative error implies that the result was lower and a positive error implies that the result was higher than the reference.

Simulation case	$\theta_{Avionics1}$	$\theta_{Avionics2}$	$\theta_{Avionics3}$	$\theta_{IsolatedPipe}$	θ_{Outlet}	Total error
Reference	241.6 °C	191.4 °C	67.1 °C	217.4 °C	142.9 °C	0 °C
RANS _{V1}	+15.3 °C	+2.9 °C	-3.8 °C	-0.9 °C	+2.0 °C	24.9 °C
RANS _{V2}	+4.1 °C	-16.6 °C	-0.1 °C	-5.2 °C	+6.3 °C	32.3 °C
RANS _{V3}	+25.8 °C	-0.2 °C	+15.8 °C	+4.4 °C	+13.5 °C	59.7 °C
RANS _{V4}	-4.2 °C	-8.8 °C	+11.3 °C	+1.6 °C	+12.3 °C	38.2 °C
RANS _{V5}	+28.0 °C	+21.5 °C	-6.8 °C	-1.7 °C	+1.2 °C	59.2 °C
RANS _{V6}	+7.6 °C	-8.7 °C	-0.6 °C	-8.6 °C	+3.8 °C	29.3 °C
RANS _{V7}	+10.3 °C	+0.9 °C	-3.6 °C	-3.3 °C	+2.0 °C	20.1 °C
RANS _{V8}	+13.8 °C	-0.8 °C	-1.7 °C	-1.2 °C	+3.0 °C	20.5 °C
RANS _{V9}	+14.2 °C	+2.2 °C	-2.3 °C	-2.2 °C	+3.0 °C	23.9 °C
RANS _{V10}	+8.0 °C	+0.8 °C	-2.2 °C	-1.9 °C	+3.1 °C	16.0 °C

The heat transfer from the fluid to parts in the domain is shown in Table 4.4. Just as in the text discussing the temperatures in Table 4.3, the total error values in Table 4.4 were obtained by summing the absolute values of the errors for each simulation.

The simulation with the smallest total error in the table is the RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model. The simulations with prism layers predict heat transfer values that are much closer to the reference result compared to the simulations without prism layers. This result is expected as a mesh with prism layers would capture the thermal boundary layer more accurately than a mesh without prism layers. Comparing the RANS simulations using the SST $k - \omega$ turbulence model on a polyhedral mesh with 11 and 20 prism layers, it is noted that the predictions of the simulation with 20 prism layers contain more errors than the simulation with only 11 prism layers. This suggests that the optimal number of prism layers for the simulations is somewhere below 20.

From the total error of the results in Table 4.4, the SST $k - \omega$ turbulence model seems to perform better on the trimmed mesh than on the polyhedral mesh, which is the opposite of what is seen in Table 4.3. The Lag EB $k - \varepsilon$ turbulence model seems to perform better on the trimmed mesh than on the polyhedral mesh for predicting heat transfer. This is noted to be true also for predicting the temperatures in Table 4.3. The simulation with Liou's AUSM+ inviscid flux scheme did not predict heat transfer results as well as the simulation using the Weiss and Smith Roe's inviscid flux scheme.

Table 4.4: Mean heat transfer from the fluid to the different parts inside the bay and to the outlet of the bay of the SRH simulation. The values for the RANS simulations are shown as errors compared to the reference simulation, where a negative error implies that the result was lower and a positive error implies that the result was higher than the reference.

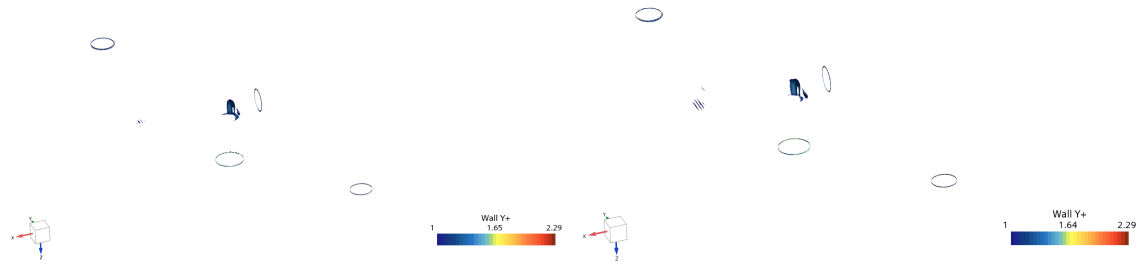
Simulation case	$HT_{AvionicsPipe}$	$HT_{IsolatedPipe}$	HT_{Outlet}	$HT_{PipeWithValve}$	Total error
Reference	-266.44 W	-298.29 W	5691.36 W	-12.54 W	0 W
RANS _{V1}	+6.77 W	-1.37 W	-105.99 W	-21.17 W	135.30 W
RANS _{V2}	-35.46 W	-7.26 W	-149.99 W	-13.42 W	206.13 W
RANS _{V3}	-206.45 W	+5.82 W	+175.01 W	+5.97 W	393.75 W
RANS _{V4}	-195.2 W	+2.18 W	+172.57 W	+15.43 W	385.38 W
RANS _{V5}	+12.16 W	-2.38 W	-98.9 W	-15.59 W	129.03 W
RANS _{V6}	-25.47 W	-11.01 W	+51.5 W	-23.51 W	111.49 W
RANS _{V7}	+6.53 W	-4.4 W	-103.97 W	-20.35 W	135.25 W
RANS _{V8}	-9.69 W	-2.05 W	-53.86 W	-19.59 W	85.19 W
RANS _{V9}	+3.63 W	-3.53 W	-161.56 W	-20.65 W	189.37 W
RANS _{V10}	-8.1 W	2.64 W	-118.81 W	-16.29 W	145.84 W

4.3.3 Comparison of the fluid fields

All the simulations with prism layers in the mesh, Figures 4.14, 4.15, 4.17, 4.18 and 4.19, had a maximum wall Y^+ below 3, where the great majority of the cells have a wall Y^+ below 1. The regions with a wall Y^+ above 1 in these simulations are either impingement zones, the ventilation hole, or regions close to the inlet or outlet boundaries. The simulations based on meshes without prism layers, Figure 4.16, had a wall Y^+ above 1 in nearly the entire domain, where a large region near the isolated pipe in the aft bay had a y^+ above 20 in both simulations.

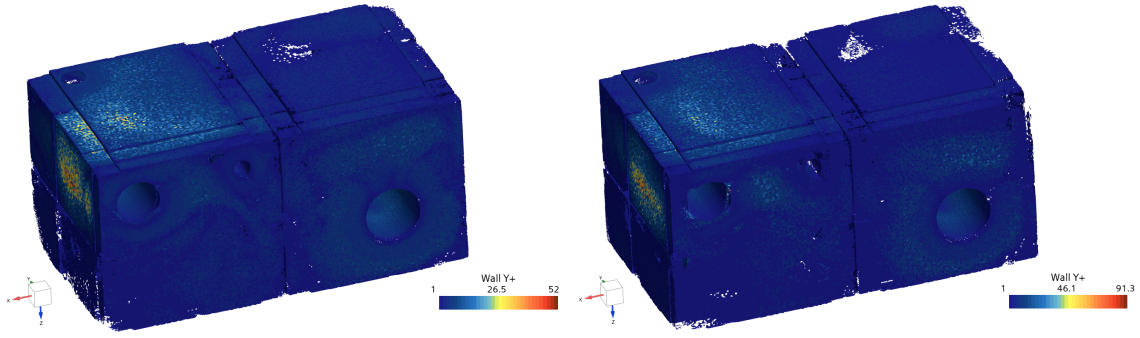


Figure 4.14: Wall Y^+ values of the reference simulation. Only wall Y^+ values above 1 are shown in the figure.



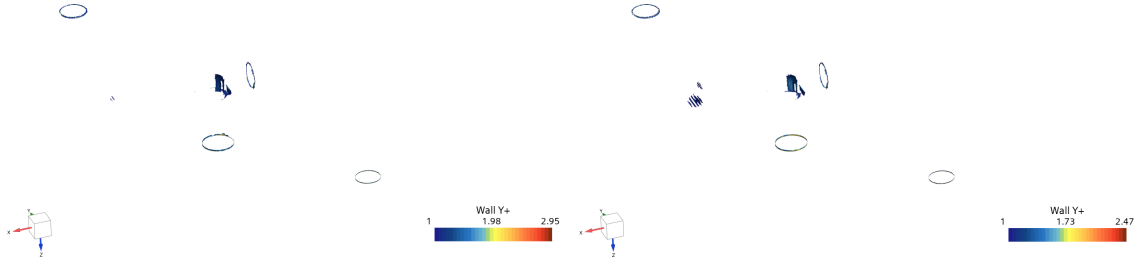
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the Lag EB $k - \epsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.15: Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.



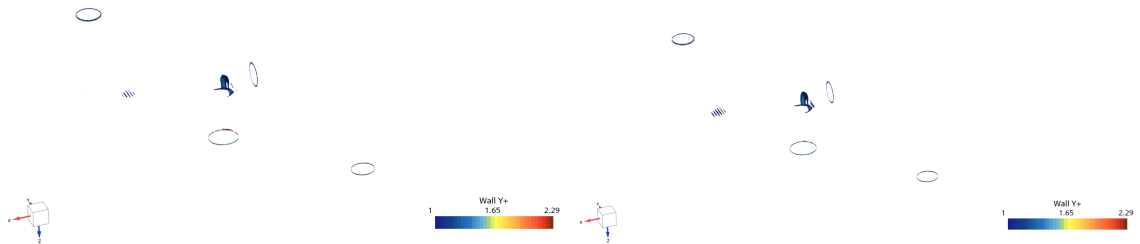
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers. (b) RANS simulation using the Lag EB $k - \epsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.16: Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.



(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers. (b) RANS simulation using the Lag EB $k - \epsilon$ turbulence model and the trimmed mesh with 11 prism layers.

Figure 4.17: Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.18: Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.

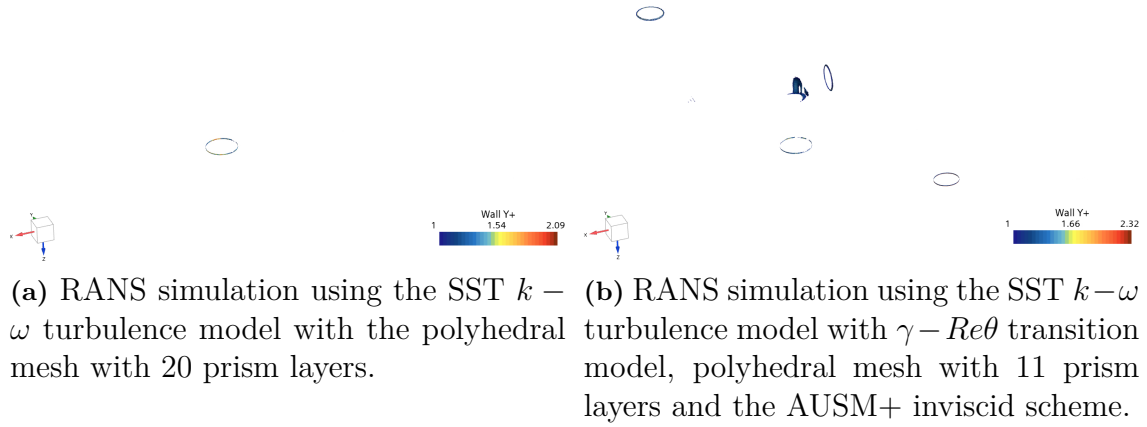


Figure 4.19: Wall Y^+ values of the RANS simulations. Only wall Y^+ values above 1 are shown in the figure.

A cross-section of the time-averaged velocity field of the SRH simulation is shown in Figure 4.20 and the mean velocity fields of the RANS simulations are shown in Figures 4.21-4.25. In the forward bay, the wake behind the avionics pipe seen in the reference simulation (Figure 4.20) has clearly separated from the pipe. This is a behaviour that none of the RANS simulations capture accurately, however, the Lag EB $k - \varepsilon$ turbulence model (Figures 4.21b, 4.22b, 4.23b) captures this separation better than the SST $k - \omega$ turbulence model (Figures 4.21a, 4.22a, 4.23a, 4.24a, 4.24b, 4.25a & 4.25b), although it is still far off from the reference. A noticeable difference in the velocity magnitude and flow shape between the RANS simulations using prism layers (Figures 4.22a & 4.22b) and the ones that do not, can be seen in the jet of air passing through the ventilation hole. The shape of the wake from the jet is best predicted by the RANS simulations using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition models and 11 prism layers in the mesh (Figures 4.24b & 4.25b). The AUSM+ inviscid flux scheme, Figure 4.25b, seems to predict a faster decrease of the velocity in the jet than the Roe inviscid flux scheme, Figure 4.24b and also predicts a slightly tighter wake from the jet, with more separation behind the isolated pipe in the aft bay.

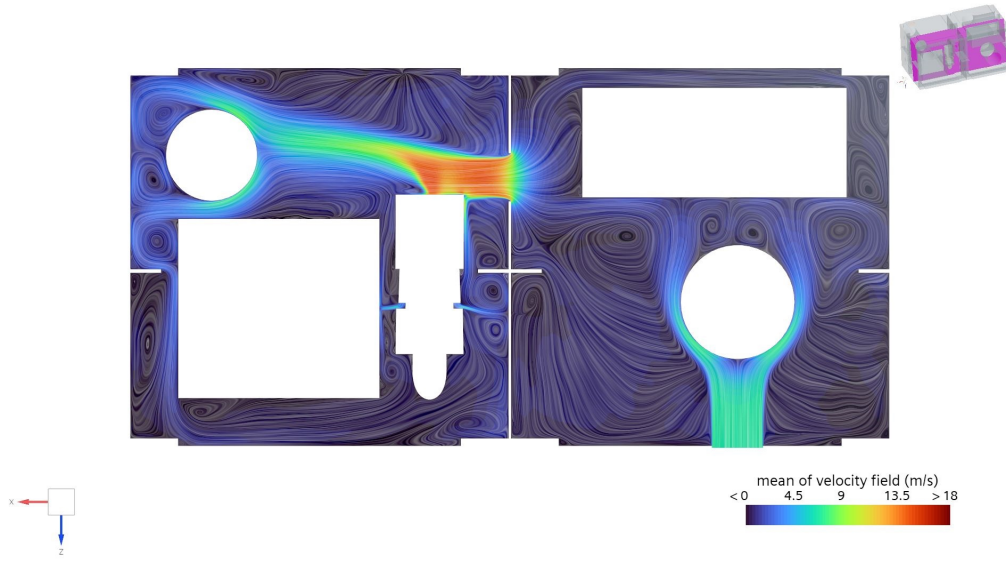
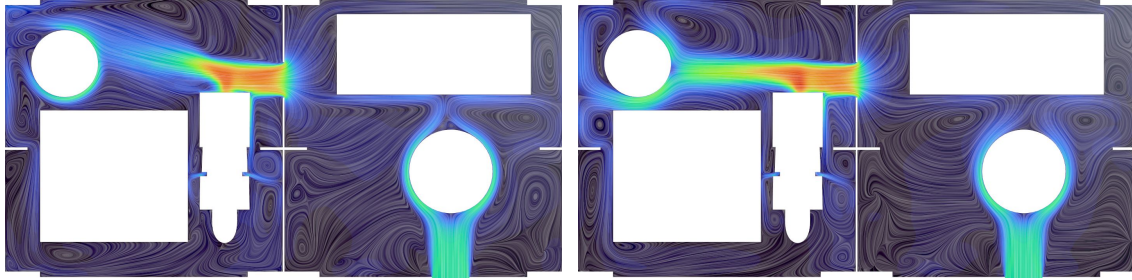
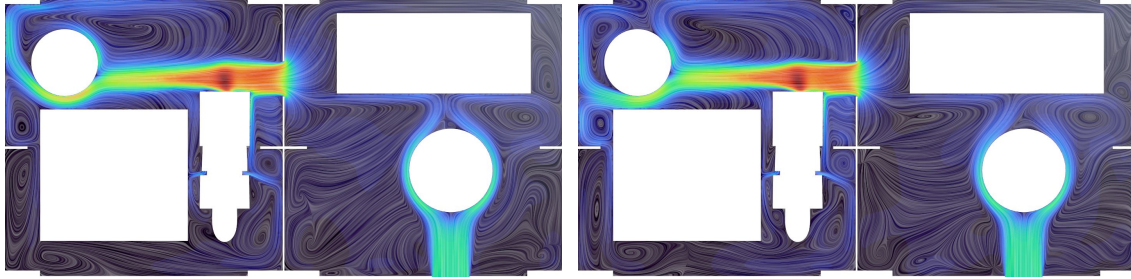


Figure 4.20: Mean velocity field of the reference simulation shown at the right cross-section.



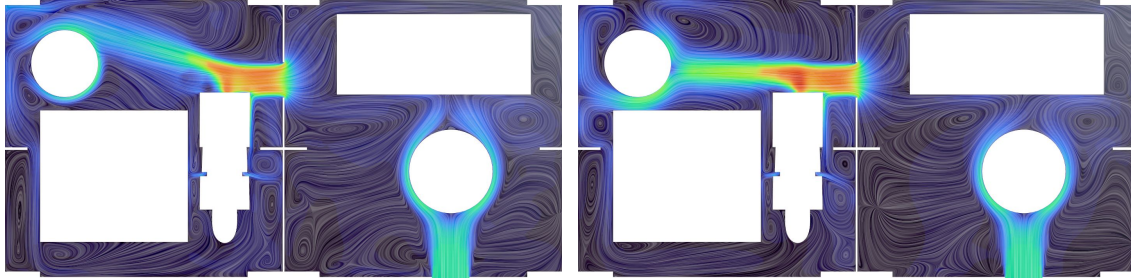
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.21: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.



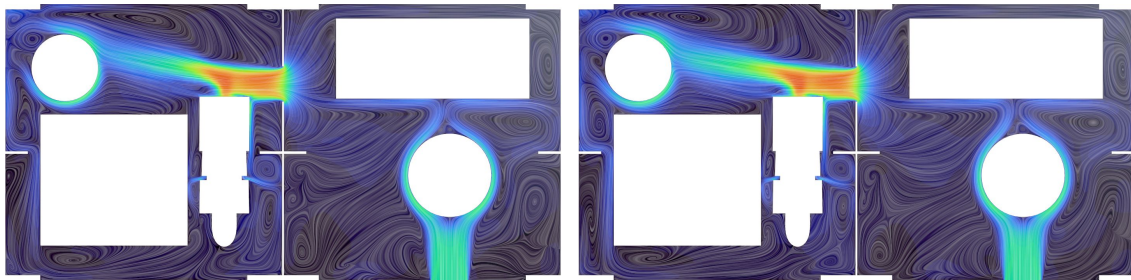
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers. (b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.22: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.



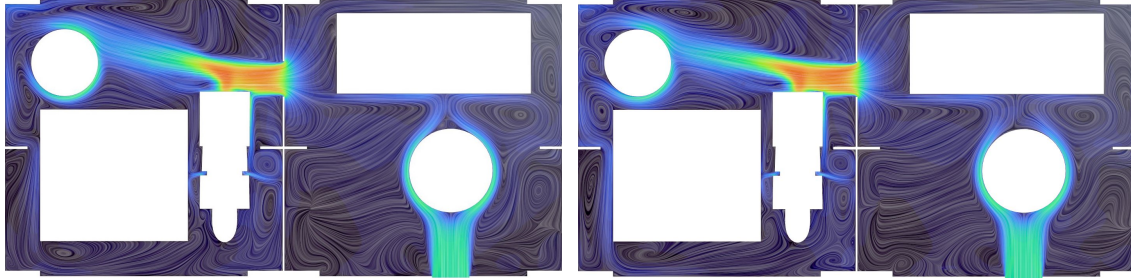
(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers. (b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the trimmed mesh with 11 prism layers.

Figure 4.23: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.24: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the polyhedral mesh with 20 prism layers. (b) RANS simulation using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme.

Figure 4.25: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.20.

The time-averaged temperature field of the reference simulation is shown in Figure 4.26. The mean temperature fields of the RANS simulations are shown in Figures 4.27-4.31. A general trend observed is that the RANS simulations based on meshes without prism layers (Figures 4.28a & 4.28b) severely overpredict the temperature in the fluid. The worst case of overpredicted temperature is seen in Figure 4.28a, which shows the RANS simulation utilizing the SST $k - \omega$ model and the polyhedral mesh without prism layers. For the RANS simulations based on a mesh with prism layers, the simulations utilizing the SST $k - \omega$ turbulence model (Figures 4.27a, 4.29a, 4.30a, 4.30b, 4.31a and 4.31b) show better prediction of the temperature in the aft bay than the simulations utilizing the Lag EB $k - \varepsilon$ model (Figures 4.27b & 4.29b).

The RANS simulation that used the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model with the AUSM+ inviscid flux scheme (Figure 4.31b) shows the most similar jet flow into the aft bay and the most similar temperature field in the aft bay, compared to the reference simulation. However, the temperature close to the forward wall of avionics 1 is overpredicted in this simulation. The most significant differences between the Roe and AUSM+ inviscid flux schemes, Figures 4.30b and 4.31b, respectively, are that the simulation with the AUSM+ inviscid flux scheme predicts the forward high temperature region from the leakage closer to the reference than the simulation with the Roe inviscid flux scheme. The AUSM+ scheme also predicts the temperature of the thermal boundary layer along the aft wall of avionics 2 closer to the reference than the Roe scheme. The overall temperature in the aft bay is also better predicted by the AUSM+ scheme than it is by the Roe scheme.

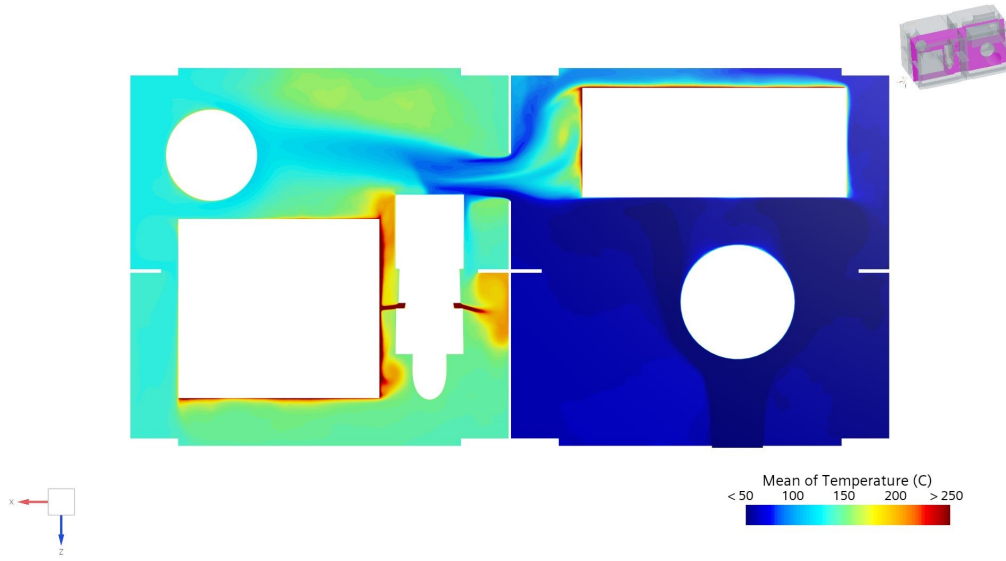
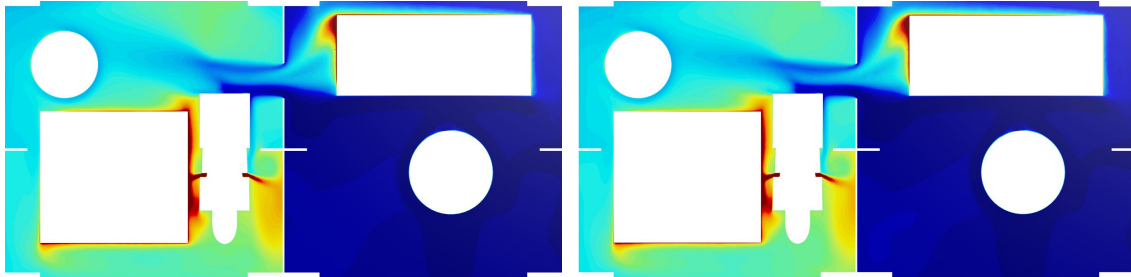
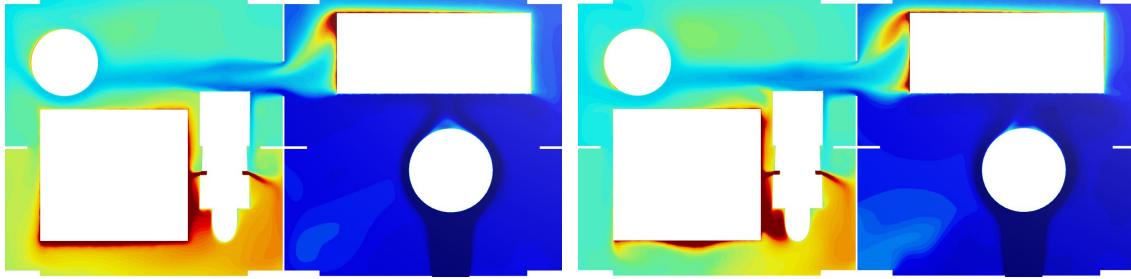


Figure 4.26: Mean temperature field of the reference simulation shown at the right cross-section.



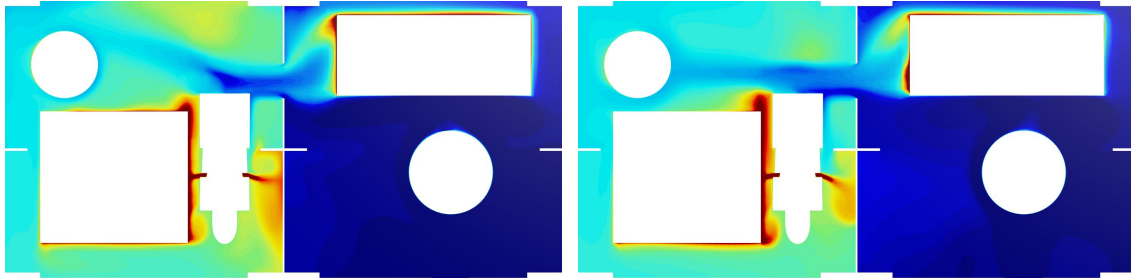
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.27: Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.



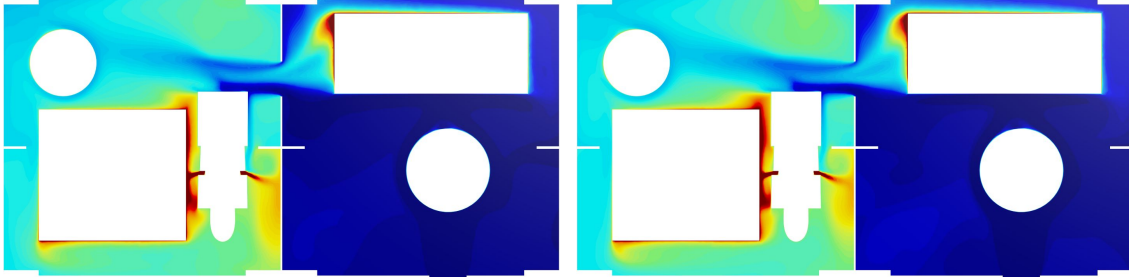
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers. (b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.28: Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.



(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers. (b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the trimmed mesh with 11 prism layers.

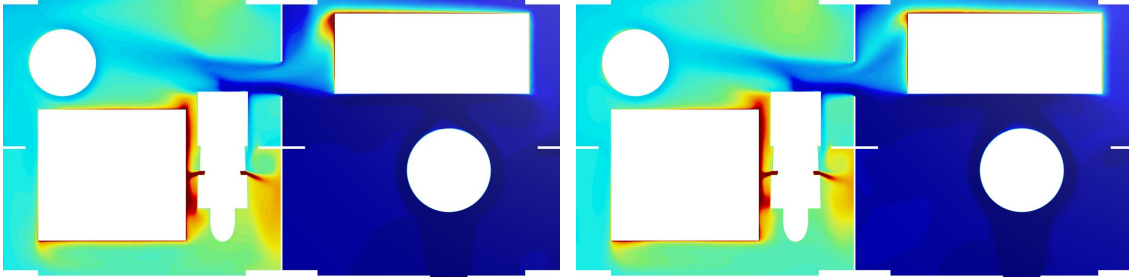
Figure 4.29: Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.



(a) RANS simulation using the SST $k-\omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers.

(b) RANS simulation using the SST $k-\omega$ turbulence model with the $\gamma-Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.30: Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.



(a) RANS simulation using the SST $k-\omega$ turbulence model with the polyhedral mesh with 20 prism layers.

(b) RANS simulation using the SST $k-\omega$ turbulence model with $\gamma-Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme.

Figure 4.31: Temperature fields of the RANS simulations on a cross-sectional plane. The temperature color bar and the location of the cross-sectional plane are found in 4.26.

The mean velocity field of the reference simulation is shown in Figure 4.32. The right side, bottom side in the figure, of the jet from the ventilation hole between the forward and aft bay hits the pipe with the leaking valve, which generates an adverse pressure gradient that causes the jet to curve slightly to the left, upwards in the figure. The flow from the jet is seen to spread along the isolated pipe in the aft bay, and some recirculation regions are visible aft of the pipe.

All of the RANS simulations, Figures 4.33-4.37, capture the adverse pressure gradient of the pipe with the leaking valve. The predicted behaviour of the jet, however, changes widely between the RANS simulations. The simulations based on a mesh with prism layers and using SST $k - \omega$ model, Figures 4.33a, 4.35a, 4.36a, 4.36b, 4.37a and 4.37b, predict the shape of the jet closer to the reference simulation than the simulations with the Lag EB $k - \varepsilon$ model, Figures 4.33b & 4.35b, do. The Lag EB $k - \varepsilon$ model predicts a large wake behind the pipe with the leaking valve, which is nonexistent in the results from the reference simulation.

The simulations without prism layers, Figures 4.34a & 4.34b, show a straighter jet with a large wake from the flow around the pipe with the leaking valve that connects to the wake from the jet itself. These simulations also predict an increase in the velocity of the jet along the wall of the pipe compared to the other simulations. This effect is most likely the result of a lack of small cells along the wall of the pipe, resulting in the boundary layer along the pipe not being captured. The simulations with the SST $k - \omega$ model on the trimmed mesh, Figure 4.35a, predict the angle of the jet better than the other RANS simulations, but it also overpredicts the velocity aft of the isolated pipe in the aft bay and underpredicts the velocity of the jet itself compared to both the other RANS simulations and the reference simulation.

The simulation based on the polyhedral mesh with 20 prism layers, Figure 4.37a, showed an identical result to the same simulation setup with only 11 prism layers, Figure 4.33a. In other words, there was no performance increase in adding more prism layers when studying the velocity of the fluid in this cross-section. There is also little to no difference between the results from the SST $k - \omega$ simulation without a transition model, Figure 4.33a, and the SST $k - \omega$ simulations with transition models, Figures 4.36a & 4.36b. The simulation with the AUSM+ inviscid flux scheme, Figure 4.37b, did not show any improved velocity predictions compared to the simulation with Roe's inviscid flux scheme, Figure 4.36b, and the reference simulation, Figure 4.32.

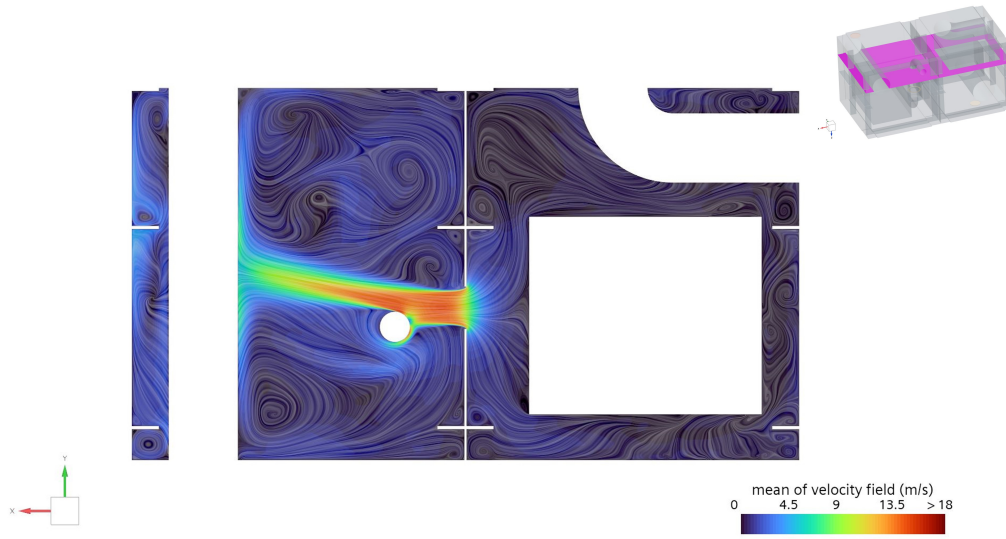
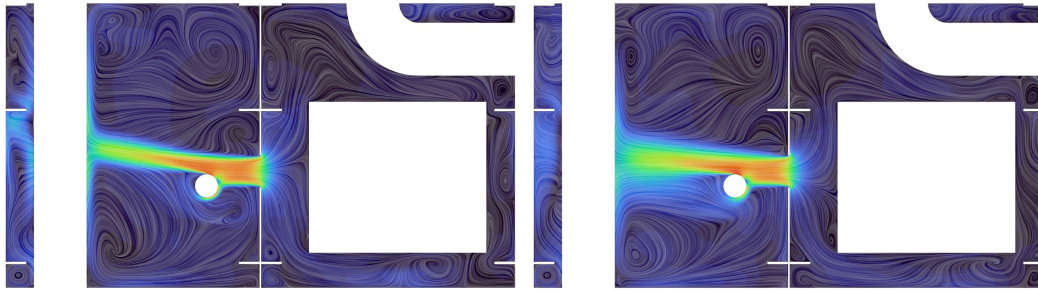
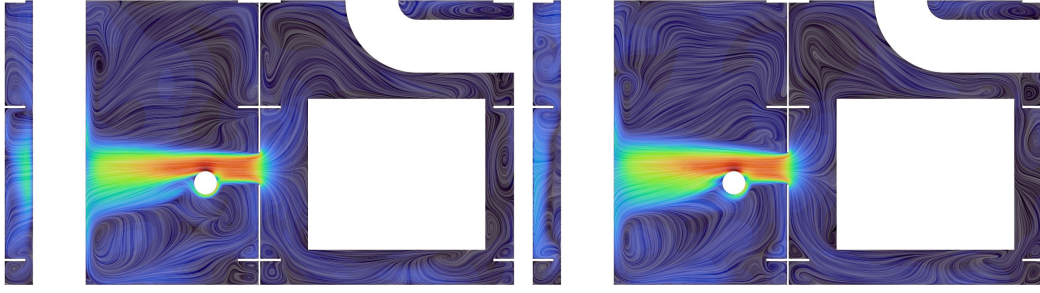


Figure 4.32: Mean velocity field of the reference simulation shown at the top middle cross-section.



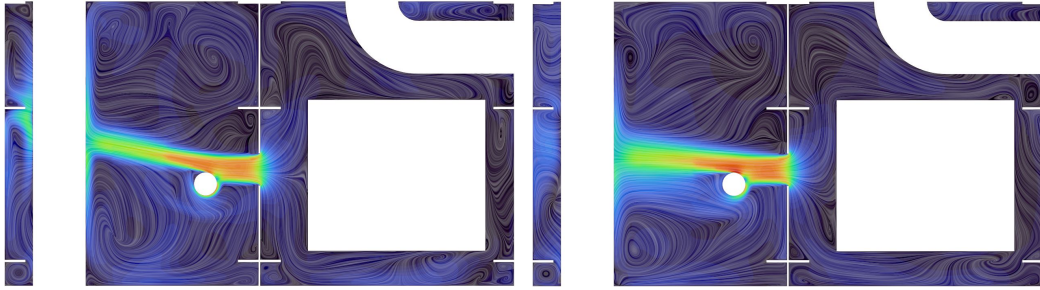
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the Lagrangian EB $k - \epsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.33: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.



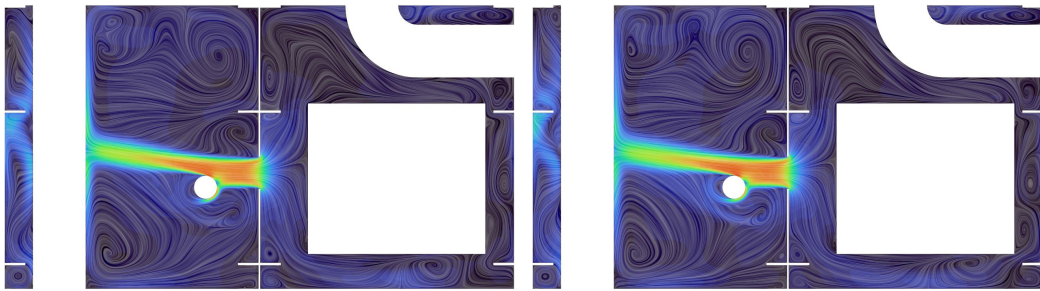
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers. (b) RANS simulation using the Lagrangian EB $k - \varepsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.34: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.



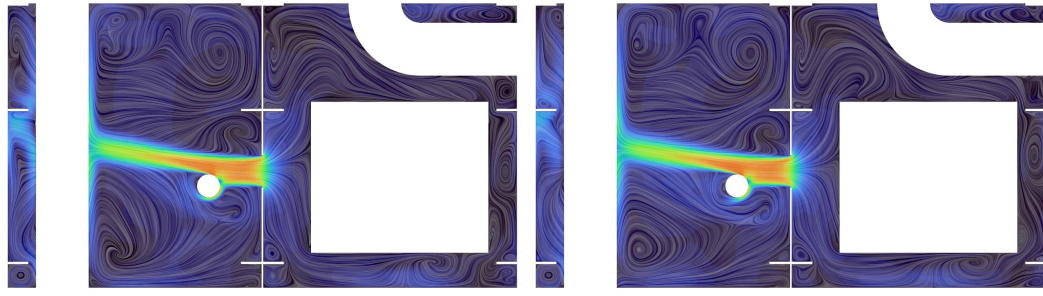
(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers. (b) RANS simulation using the Lagrangian EB $k - \varepsilon$ turbulence model and the trimmed mesh with 11 prism layers.

Figure 4.35: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.36: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the polyhedral mesh with 20 prism layers. (b) RANS simulation using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme.

Figure 4.37: Velocity fields of the RANS simulations on a cross-sectional plane. The velocity color bar and the location of the cross-sectional plane are found in 4.32.

Figures 4.38-4.43 show the mean temperature field on a cross-section located roughly in the middle of the ventilation hole, normal to the z -axis. The cross-section is viewed from the top. Studying the temperature in the cross-section shown in Figure 4.38, the reference simulation shows that the cold jet is tight and does not mix with the surrounding air in the aft bay until the jet hits the isolated pipe, where the cooling air spreads out along the pipe. All the RANS simulations show an overall colder right side, below the jet in the figures, of the aft bay compared to the reference.

Simulation using the polyhedral mesh without prism layers, Figures 4.40a & 4.40b, shows an expanding wake from the jet. The cooling jet from the ventilation hole is almost normal to the ventilation hole, which differs from the reference simulation, Figure 4.38. This could be the result of the jet hitting the pipe with the leaking valve, causing some separation behind the small pipe, resulting in the increased size of the wake. The effect is also visible in Figure 4.41b, showing the simulation using the Lag EB $k - \varepsilon$ turbulence model on a trimmed mesh. However, in this simulation, the wake behind the pipe with the leaking valve is far more pronounced. In general, this phenomenon might be caused by poorly placed cells in the mesh.

The simulation using the SST $k - \omega$ turbulence model on the trimmed cell mesh with prism layers, Figure 4.41a shows a colder jet flow than both the reference simulation and the other simulations using the SST $k - \omega$ turbulence model on a polyhedral mesh with prism layers, Figures 4.39a, 4.42a, 4.42b, 4.43a and 4.43b. Both simulations utilizing the trimmed mesh, Figures 4.41a & 4.41b, underpredict the temperature in the forward bay slightly compared to the simulations based on the polyhedral mesh with prism layers, suggesting that the polyhedral mesh is better for these simulations. It is also noted that, as previously discussed, the simulations based on meshes without prism layers, Figures 4.40a & 4.40b, tend to overpredict the temperature, which is clearly visible in the forward bay.

It is difficult to say which simulation setup predicts the temperature field the best from this cross-section, but the RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme, Figure 4.43b, seems like the best candidate. The simulation predicts the temperature of the bulk fluid in the forward and aft bay closest to the reference simulation. The flow around the pipe with the leaking valve and the lack of a cold wake from the pipe is comparable to the reference simulation.

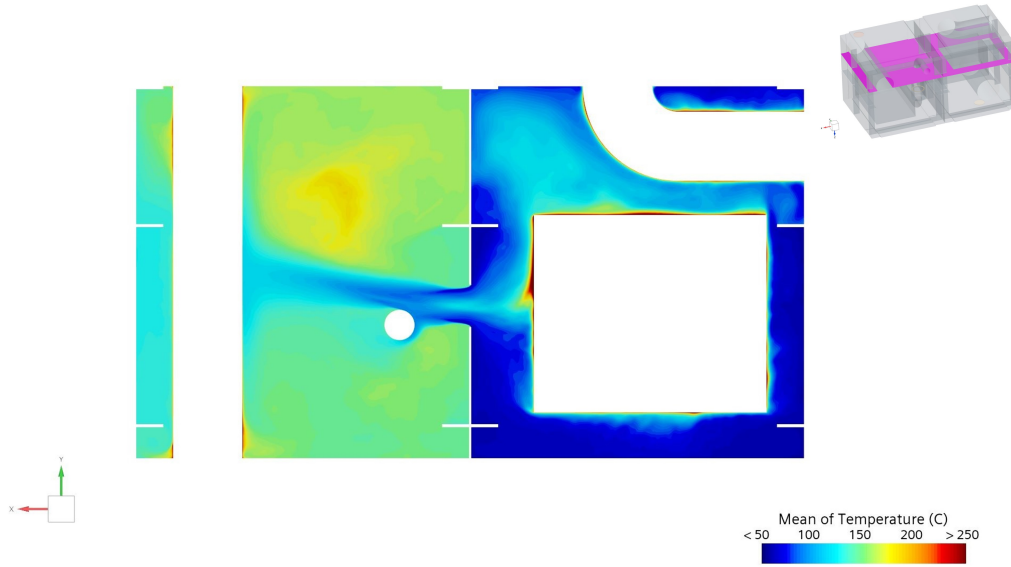
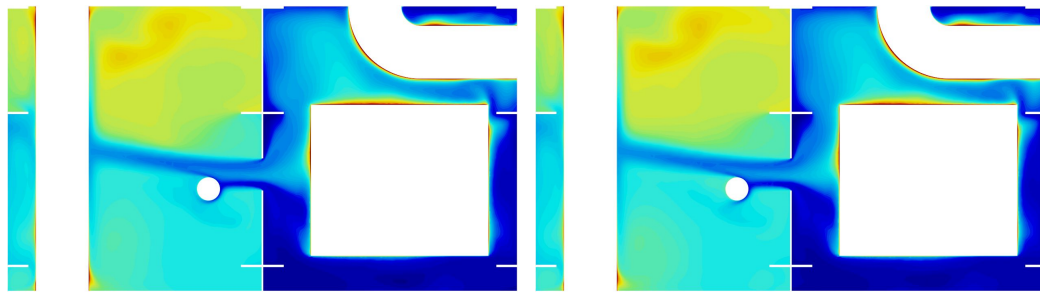
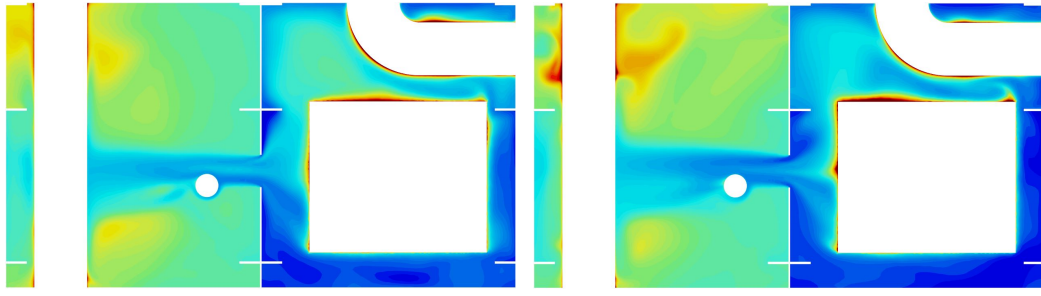


Figure 4.38: Mean temperature field of the reference simulation shown at the top middle cross-section, the location of which is shown in the top right of the figure.



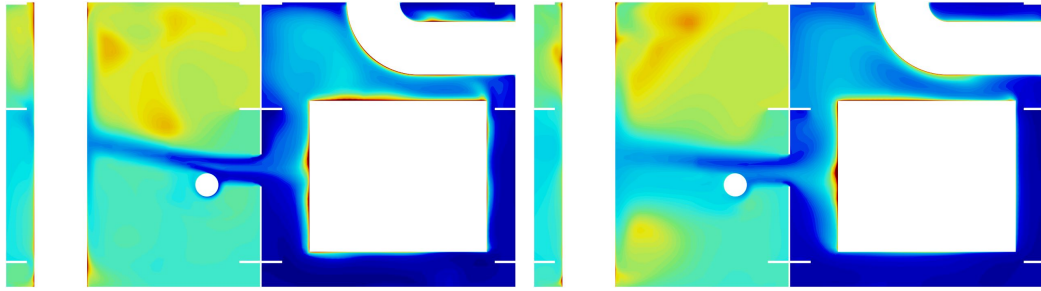
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the Lagrangian $k - \varepsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.39: Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38.



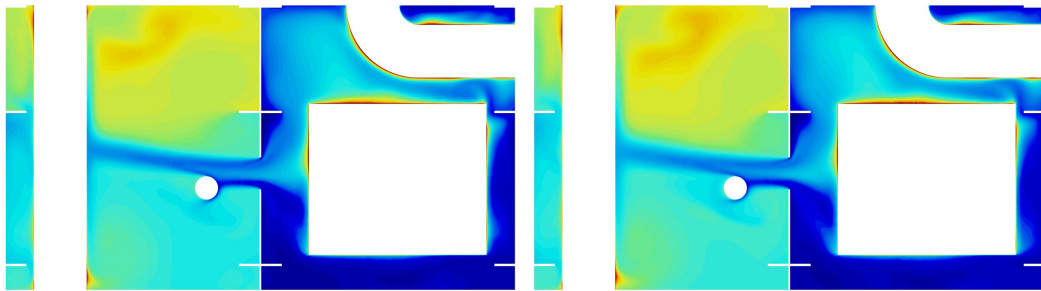
(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers. (b) RANS simulation using the Lagrangian EB $k - \epsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.40: Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38.



(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers. (b) RANS simulation using the Lagrangian EB $k - \epsilon$ turbulence model and the trimmed mesh with 11 prism layers.

Figure 4.41: Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers. (b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.42: Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38.

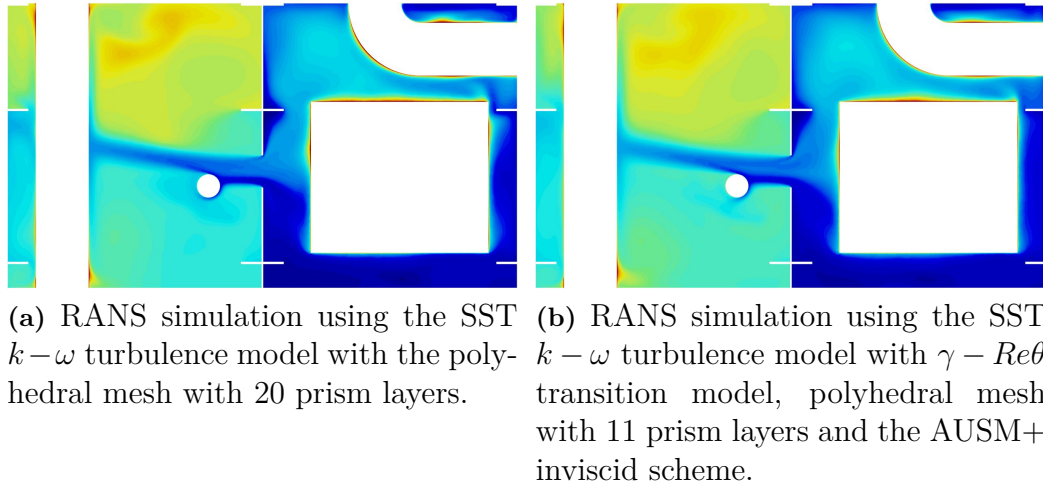


Figure 4.43: Temperature fields of the RANS simulations. The temperature color bar and the location of the cross-sectional plane are found in 4.38.

Figure 4.44 shows the isosurface at 200 °C in the aft bay of the reference simulation. Comparing the RANS simulations, Figures 4.45-4.49, to the reference simulation, it is clear that none of the RANS simulations capture the temperature field at 200 °C correctly. The simulations based on the polyhedral mesh with prism layers, Figures 4.45a, 4.48a, 4.48b, 4.49a and 4.49b, produce isosurfaces more similar to the reference simulation than the simulations based on the trimmed mesh with prism layers, Figures 4.47a & 4.47b. The simulations without prism layers, Figures 4.46a & 4.46b, show an isosurface with a large difference to the reference simulation. This is expected since the previously discussed results from the simulations without prism layers indicated a tendency for overpredicting the temperature in the fluid.

The simulation showing an isosurface closest to that of the reference simulation is the RANS simulation using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid flux discretization scheme, Figure 4.49b. The SST $k - \omega$ simulations with transition models, Figures 4.48a & 4.48b, showed to produce isosurfaces closer to the reference than the simulation without, Figure 4.45a, especially the simulation with the $\gamma - Re\theta$ transition model, Figure 4.48b.

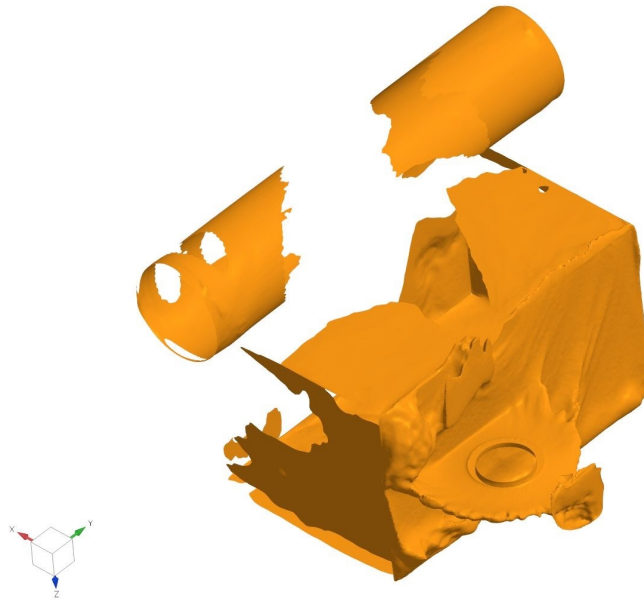
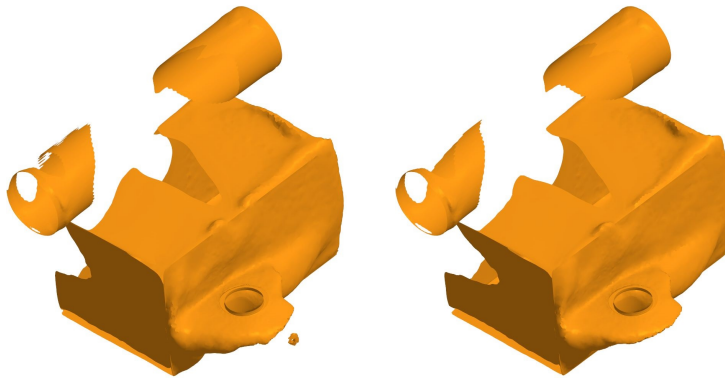


Figure 4.44: Isometric view of the isosurface of the temperature at 200 °C from the reference simulation.



(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers.

(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.45: Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.

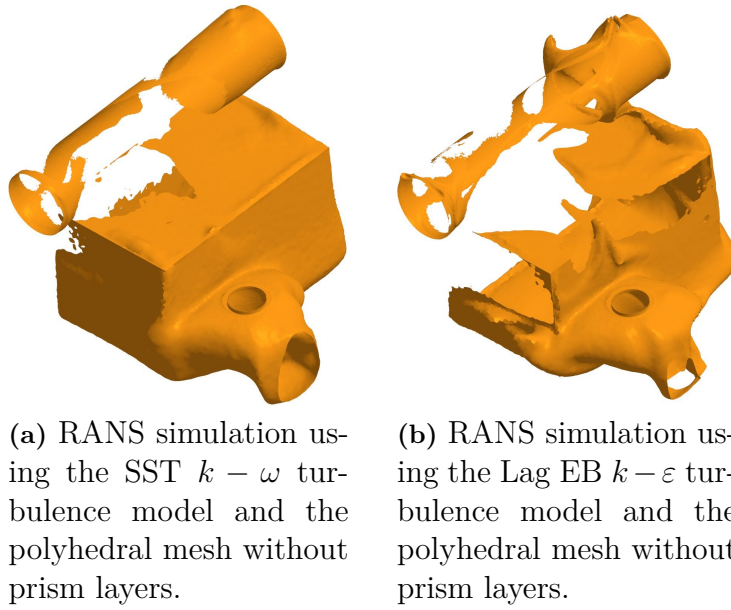


Figure 4.46: Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.

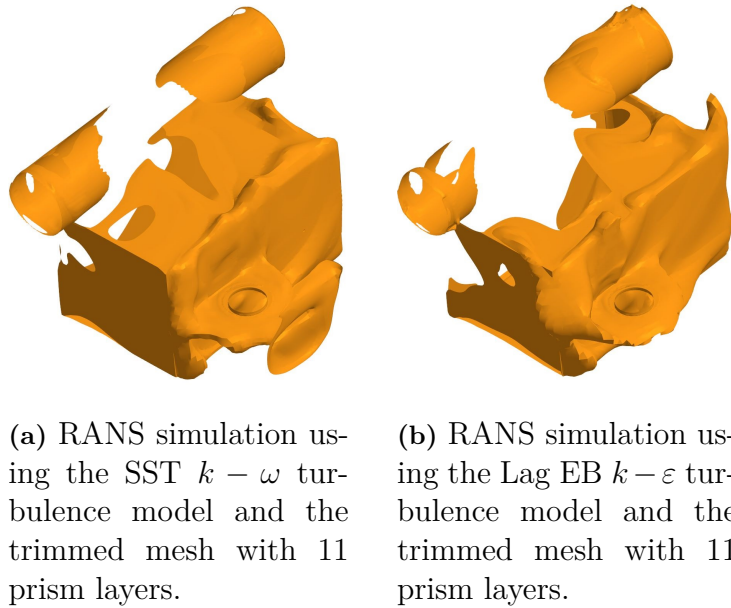
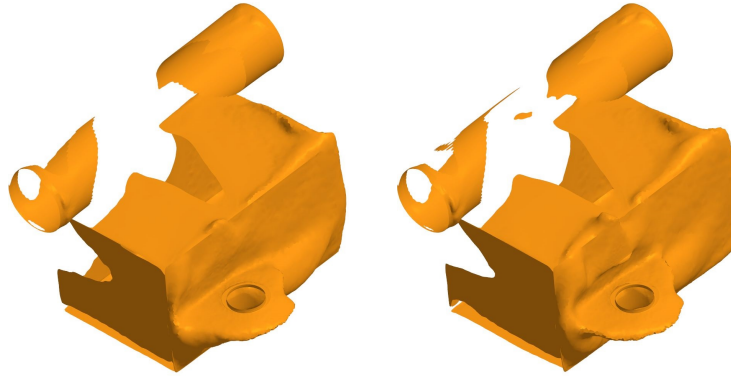


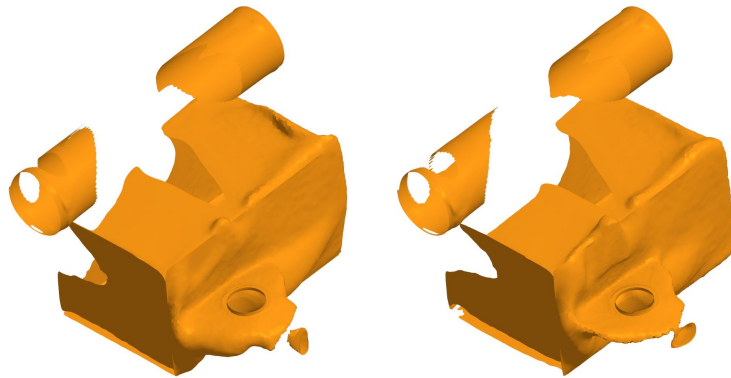
Figure 4.47: Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers.

(b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.48: Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the polyhedral mesh with 20 prism layers.

(b) RANS simulation using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme.

Figure 4.49: Isometric view of the isosurface of the temperature at 200 °C from the RANS simulations.

4.3.4 Comparison of velocity profiles

The velocity profiles along the top surface of avionics 1, created from the marked points in Figure 4.50, are shown in Figure 4.51. The ratio $\frac{2V}{V_{max}}$ for the velocity profiles was chosen as a representative ratio in the figure, and V_{max} is the maximum value of the scales in Figures 4.20 & 4.32, i.e. $V_{max} = 18$ m/s. The graph labeled SRH refers to the reference simulation. Only every other of the locations along the x -axis in Figure 4.50 is plotted to reduce the clutter in Figure 4.51.

Studying the graph of the reference simulation, the velocity profile along the x -axis clearly shows the development of a boundary layer near the wall. The boundary layer of the reference simulation is nearing separation at $x = 0.44$ as the velocity decreases from $x = 0.42$. The boundary layer does, however, stay attached along the entirety of the top surface of avionics 1 for the reference simulation.

Studying the RANS simulations, a very noticeable behaviour is observed: both simulations show a velocity greater than zero near the wall at several locations. This lack of prism layers means that the cells close to the wall are too large to capture a boundary layer properly. Still, some of the boundary layer is captured at a later location. At $x = 0.42$, for example, the velocity profiles of the simulations without prism layers compare to the reference simulation further away from the surface. This is because the boundary layer has grown thick enough for the normal-sized cells in the mesh to capture it. The RANS simulations using the SST $k - \omega$ turbulence model on the polyhedral mesh without prism layers also show a separation point at $x = 0.46$, where the velocity in the points near the wall goes to zero.

The simulation using the SST $k - \omega$ turbulence model on a trimmed mesh captures the development of the boundary layer but underpredicts the magnitude of the velocity at all locations compared to the reference simulation. The RANS simulation also predicts a separation at $x = 0.44$, earlier than the RANS simulation using the SST $k - \omega$ model and a mesh without prism layers.

The Lag EB $k - \varepsilon$ turbulence model performs slightly better on the trimmed mesh with prism layers than on the polyhedral mesh with prism layers. It captures the boundary layer development well, but captures the start of the separation slightly later than the reference. The simulations using the different SST $k - \omega$ models on the polyhedral mesh with prism layers show very similar velocity profiles. The increased number of 20 prism layers does not seem to provide any meaningful increase in predicting the boundary layer along the top surface of avionics 1. The AUSM+ inviscid flux discretization scheme performed a lot worse at the separation point of $x = 0.46$ than Roe's scheme, suggesting that Roe's scheme should be used for simulations where the boundary layer is of interest.

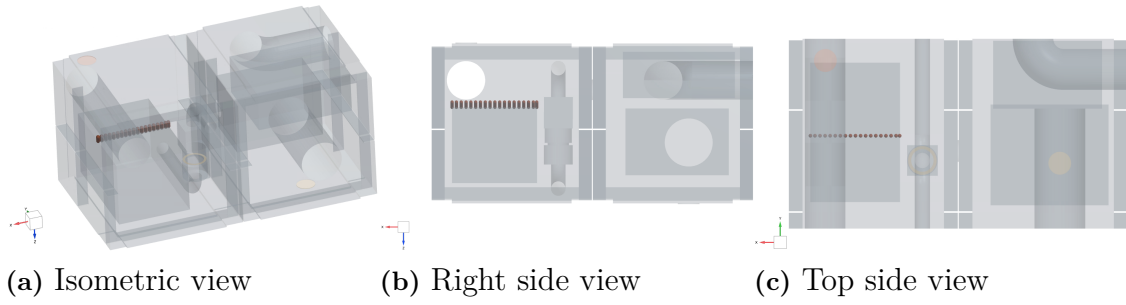


Figure 4.50: Figures showing the locations of the measured data presented in Figure 4.51.

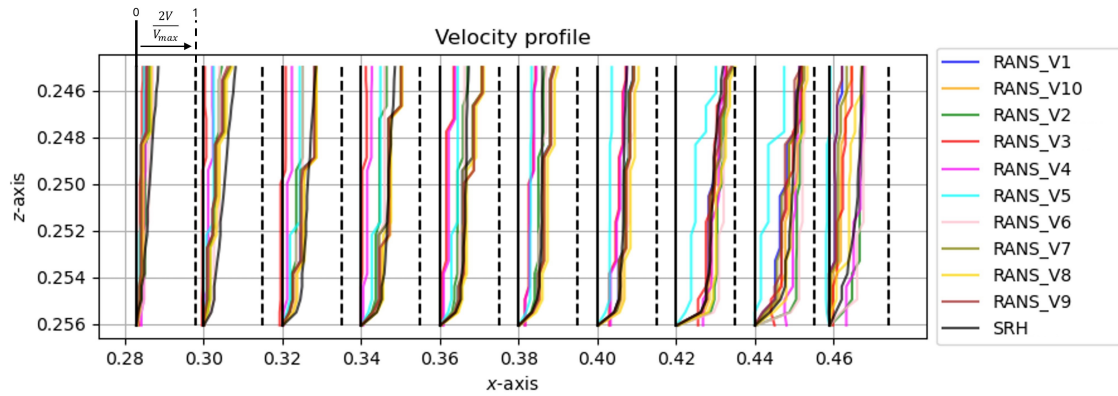


Figure 4.51: Figure showing the velocity profiles of the simulations along the top surface of avionics 1. The figure only shows the profile of the x -component of the velocity, since this is the dominating component due to the jet.

The velocity profiles inside the jet, created from the points shown in Figure 4.52, are shown in Figure 4.53. The ratio $\frac{V}{V_{max}}$ is the ratio of local velocity to the maximum velocity, $V_{max} = 18$. Only every third of the locations along the x -axis in Figure 4.52 is plotted to reduce the clutter in Figure 4.53.

The general trend of the velocity profiles in Figure 4.53 is that the simulations using the SST $k - \omega$ turbulence model on a mesh with prism layers predict a profile closer to the reference than any of the other RANS simulations. This could be due to the fact that the reference simulation was also run using the SST $k - \omega$ turbulence model. All the simulations using the Lag EB $k - \varepsilon$ turbulence model show similar velocity profiles in the wake. The reason to why even the simulation without prism layers in the mesh shows a profile close to the simulations with prism layers in the mesh is most likely because at around $x = 0.27$, the jet or wake of the jet is far enough away from any wall that the prism layers don't increase the accuracy of the prediction. When the flow hits the wall of the isolated pipe, the velocity has already reduced significantly.

The RANS simulations, which predicted velocity profiles closest to the reference simulation, are the simulations using the SST $k - \omega$ turbulence model with a transition model on a mesh with prism layers.

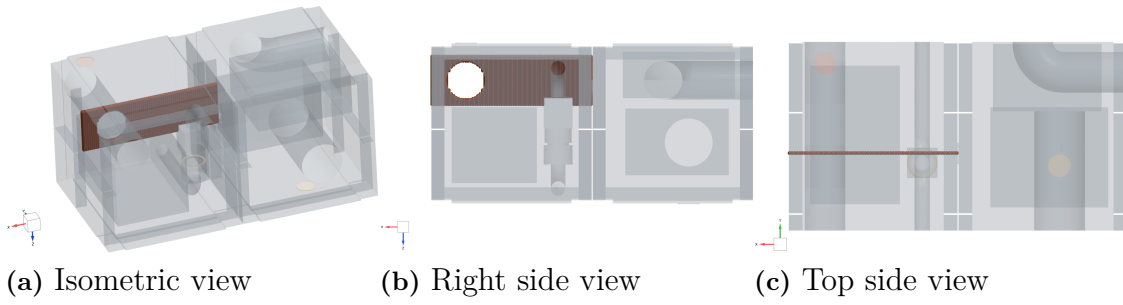


Figure 4.52: Figures showing the locations of the measured data presented in Figure 4.53.

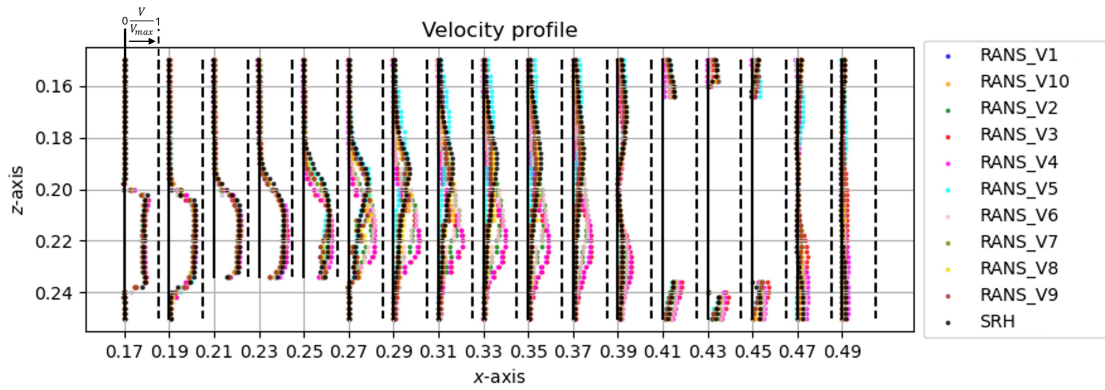


Figure 4.53: Figure showing the velocity profiles of the simulations along the top surface of avionics 1. The figure only shows the profile of the x -component of the velocity, since this is the dominating component due to the jet.

4.3.5 Comparison of surface temperatures

Figure 4.54 shows the temperature at the top of avionics 1, the surface marked in the upper right of the figure. The area where the wake of the cooling jet from the ventilation hole flows along the surface is clearly visible as the cold streak in roughly the middle of the surface. A colder area is also seen on the right side, left side of the figure, at the aft of the surface, which could be the result of a recirculation region. Figures 4.55-4.59 show the different RANS simulations of the same surface.

A noticeable difference between the RANS simulations and the reference simulation is that the RANS simulations overpredict the temperature of the surface, predicting hotspots at 350 °C or above in almost all simulation setups. The simulation that shows the least amount of hotspots is the RANS simulation using the SST $k - \omega$ turbulence model on a polyhedral mesh without prism layers. The cold wake from the jet is, however, less pronounced for this simulation. It is interesting that the simulation setup, which overall predicts a higher surface-averaged temperature, also predicts fewer hotspots. However, this figure only shows one side of a six-sided object, so it is not possible to draw any conclusions from this result.

The simulations using the Lag EB $k - \varepsilon$ turbulence model on a mesh with prism layers, Figures 4.55b & 4.57b, capture the effect of the wake on the surface and the colder region at the aft of the surface better than the RANS simulations using the SST $k - \omega$ turbulence model in all forms. However, the Lag EB $k - \varepsilon$ model also predicts hot regions around or above 350 °C, which do not exist in the result from the reference simulation.

The general shape of the temperature profile on the surface of avionics 1 is better predicted by the SST $k - \omega$ simulations using a polyhedral mesh with prism layers, Figures 4.55a, 4.58a, 4.58b, 4.59a and 4.59b. The simulation using the SST $k - \omega$ turbulence model on the trimmed mesh, Figure 4.57a, severely overpredicts the hot regions close to the front and left edges, bottom and right edges in the figure, of the surface, once again suggesting that the SST $k - \omega$ turbulence model works better on a polyhedral mesh. The RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model on a polyhedral mesh with 11 prism layers and the AUSM+ inviscid flux discretization scheme, Figure 4.59b, seems to predict values that are closer to the reference simulation than any of the other investigated RANS simulations.

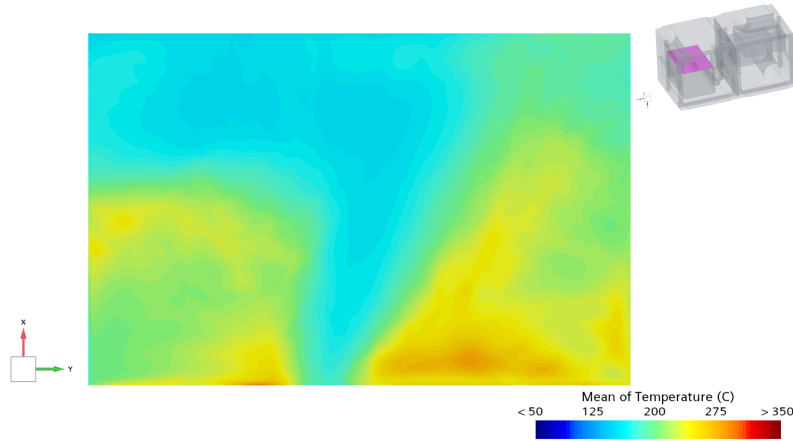
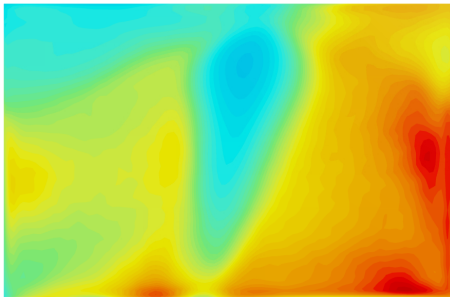
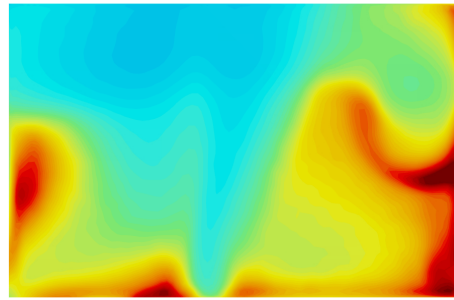


Figure 4.54: Reference simulation showing the temperature of the top surface of avionics 1. The location of the plane is shown highlighted in the top right of the figure.

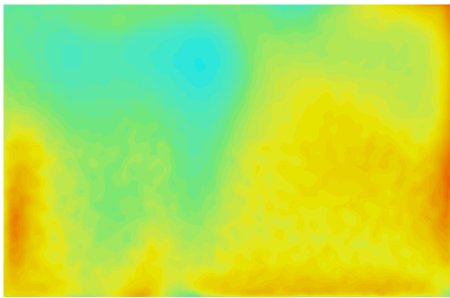


(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers.

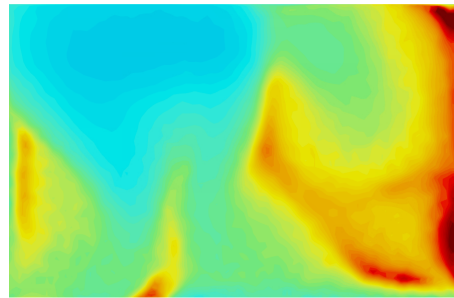


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.55: RANS simulations showing the temperature of the top surface of avionics 1.

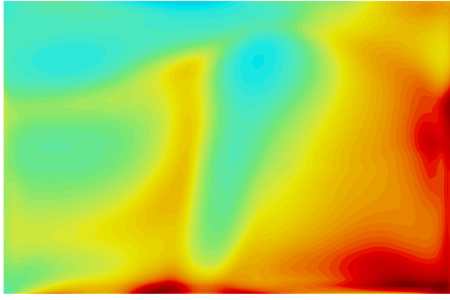


(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers.

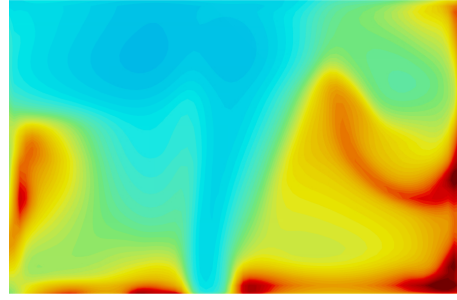


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.56: RANS simulations showing the temperature of the top surface of avionics 1.

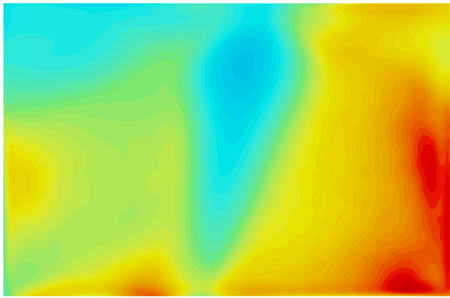


(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers.

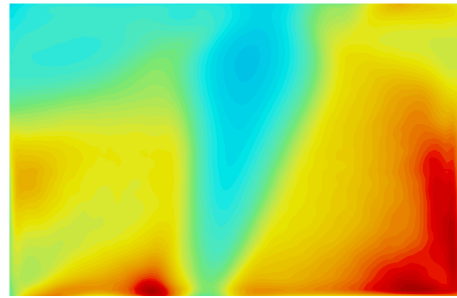


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the trimmed mesh with 11 prism layers.

Figure 4.57: RANS simulations showing the temperature of the top surface of avionics 1.

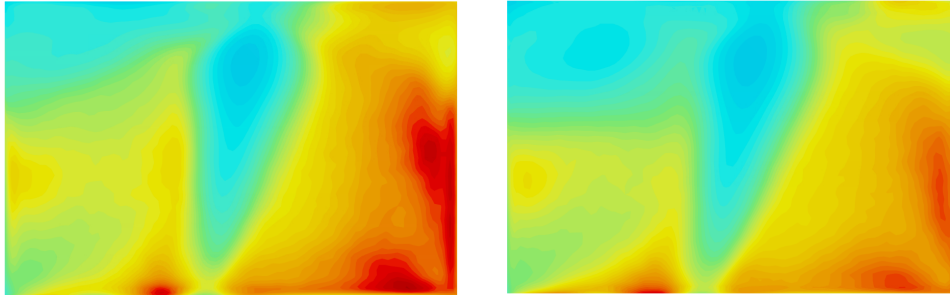


(a) RANS simulation using the SST $k - \omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers.



(b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.58: RANS simulations showing the temperature of the top surface of avionics 1.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the polyhedral mesh with 20 prism layers.

(b) RANS simulation using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme.

Figure 4.59: RANS simulations showing the temperature of the top surface of avionics 1.

The temperature of the forward surface of avionics 1 from the reference simulation is shown in Figure 4.60. The impingement zone from the leaking valve is clearly visible as the large, almost oval hot spot on the face. Another line of hotter temperature is visible almost as a circle around the previously mentioned hot spot. This could be the result of a recirculation region occurring around the impingement zone. There is also another hot region near the bottom left edge, the bottom right edge in the figure, of the surface. Once again, none of the RANS simulations, Figures 4.61-4.65, capture the impingement region of the leakage shown in the results of the reference simulation correctly.

The simulations without prism layers in the mesh, Figures 4.62a & 4.62b, do not capture any of the flow features shown in the solution of the reference simulation. Both of the simulations actually underpredict the temperature of the surface severely. This result is most likely due to the low number of cells in the region, making it hard for the solver to capture the flow properly since the surface is relatively close to the leaking valve.

Comparing the results of the SST $k - \omega$ and Lag EB $k - \varepsilon$ turbulence models on the polyhedral mesh with prism layers, the Lag EB $k - \varepsilon$ model, Figure 4.61b, showed a larger region of hot temperature on the ring from the impingement zone than the SST $k - \omega$ model, Figure 4.61a, and the reference. The Lag EB $k - \varepsilon$ model does, however, predict the curve of the region closer to the reference than the SST $k - \omega$ model. The SST $k - \omega$ model seems to predict the shape of the impingement zone closer to the reference than the Lag EB $k - \varepsilon$ model.

The simulations based on the trimmed mesh, Figures 4.63a & 4.63b, showed less curvature of the ring around the impingement zone than predicted by the reference simulation. This is most likely because the trimmed mesh is built by ordered cells,

promoting the transport of information in the Cartesian directions. The ring also covered a larger region and the region also showed a hotter temperature than the reference simulation had predicted. The Lag EB $k - \varepsilon$ turbulence model predicted a smaller region of colder temperature at the top edge of the surface, near the impingement zone, as predicted by the reference simulation. The SST $k - \omega$ turbulence model did not predict this cold region, suggesting that Lag EB $k - \varepsilon$ model works better on a trimmed mesh than the SST $k - \omega$ does.

The RANS simulation based on the polyhedral mesh with 20 prism layers, Figure 4.65a, shows a more accurate prediction of the temperature in the hot regions than the RANS simulations with only 11 prism layers, Figure 4.61a, when compared to the reference simulation. The general temperature profile of the surface is, however, identical for the two RANS simulations.

Studying the effects of the transition models for the SST $k - \omega$ model, both the simulation with the γ transition model, Figure 4.64a, and the simulation with the $\gamma - Re\theta$ transition model, Figure 4.64b, capture the flow features predicted by the reference simulation better than the simulation using no transition model, Figure 4.61a. The RANS simulation with the AUSM+ inviscid flux scheme, Figure 4.65b, performed worse than the simulation with Roe's inviscid flux scheme, Figure 4.64b. The RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model on the polyhedral mesh with 11 prism layers and Roe's inviscid flux discretization scheme performed the closest to the reference simulation of the RANS simulations.

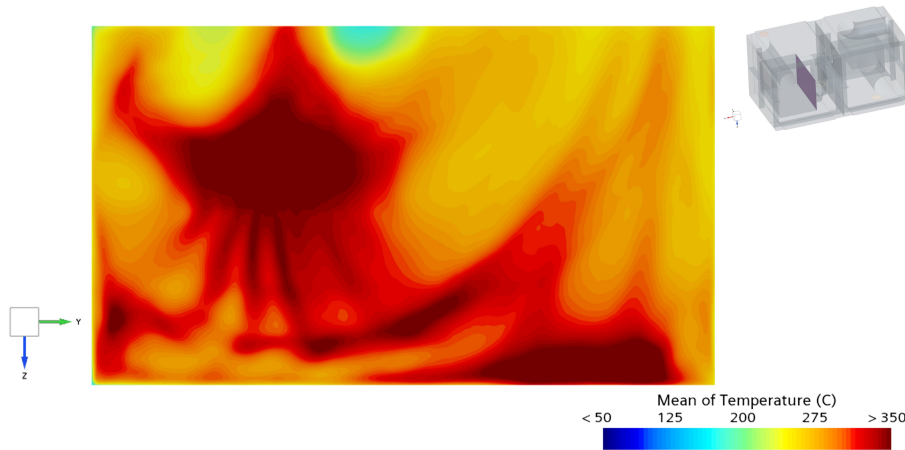
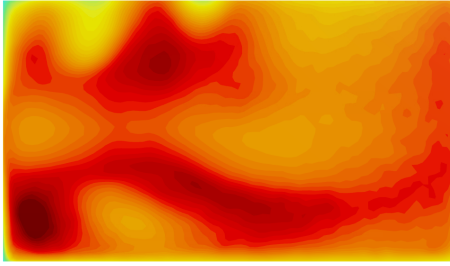
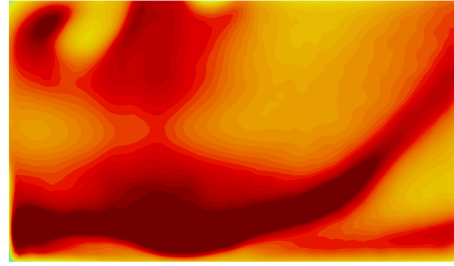


Figure 4.60: Reference simulation showing the temperature of the forward surface of avionics 1. The location of the plane is shown highlighted in the top right of the figure.

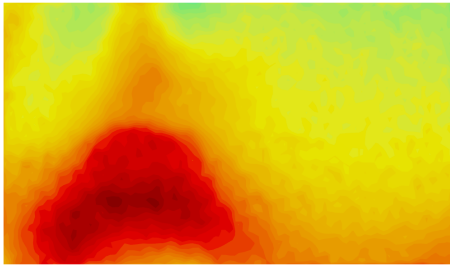


(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers.

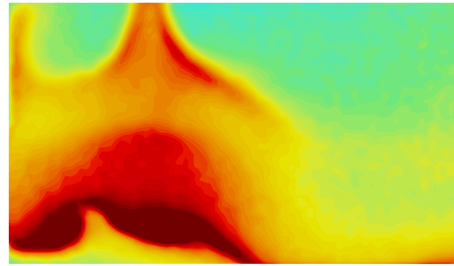


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.61: RANS simulations showing the temperature of the forward surface of avionics 1.

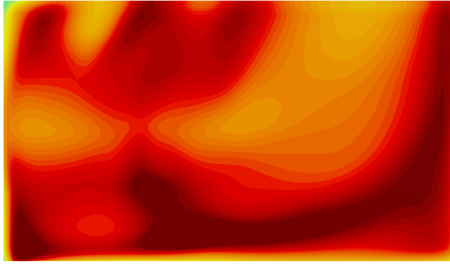


(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers.

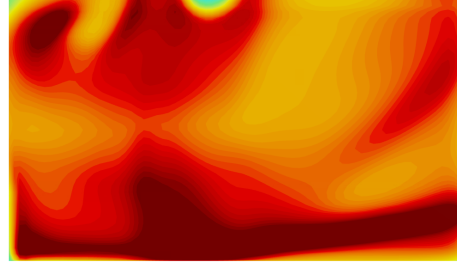


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.62: RANS simulations showing the temperature of the forward surface of avionics 1.

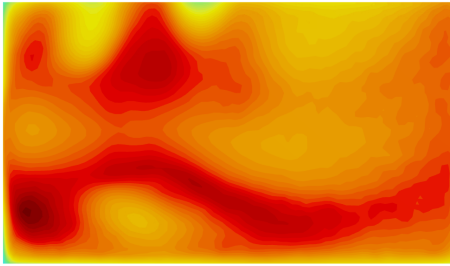


(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers.

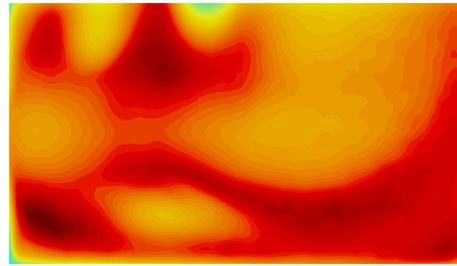


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the trimmed mesh with 11 prism layers.

Figure 4.63: RANS simulations showing the temperature of the forward surface of avionics 1.

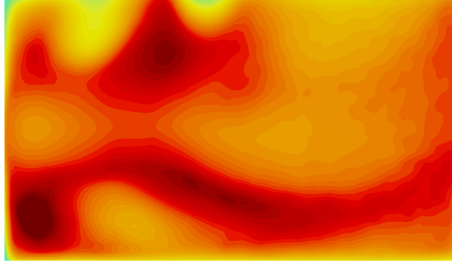


(a) RANS simulation using the SST $k - \omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers.

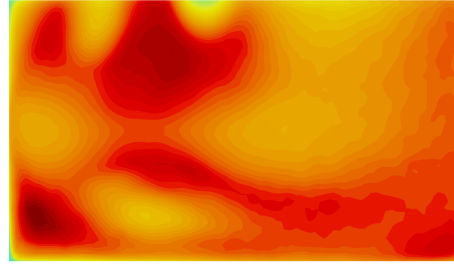


(b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.64: RANS simulations showing the temperature of the forward surface of avionics 1.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the polyhedral mesh with 20 prism layers.



(b) RANS simulation using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme.

Figure 4.65: RANS simulations showing the temperature of the forward surface of avionics 1.

Figure 4.66 shows the temperature profile of the forward-facing surface of the isolated pipe in the aft bay for the reference simulation. The region where the cold jet's wake hits the surface is clearly visible, and the cooling effect extends to the left and right of the impingement zone as the cold air moves away from the region. There are two regions at the rightmost, to the left in the figure, and leftmost, to the right in the figure, walls where the temperature is upwards of 250 °C.

Figures 4.67 - 4.71 show the temperature of the front surface of the isolated pipe in the aft bay produced by the RANS simulations. In general, the simulations with the SST $k - \omega$ turbulence model using a mesh with prism layers, Figures 4.67a, 4.69a, 4.70a, 4.70b, 4.71a and 4.71b, show an impingement zone and cooling spread closer to that of the reference simulation than the simulations using the Lag EB $k - \varepsilon$ turbulence model, Figures 4.67b, 4.68b and 4.69b, and the simulations without prism layers in the mesh, Figures 4.68a & 4.68b. Increasing the number of prism layers from 11, Figure 4.67a, to 20, Figure 4.71a did not affect the result noticeably. All the RANS simulations with prism layers underpredicted the temperature of the impingement zone, showing more cooling than the reference simulation predicted. The RANS simulation that captured the temperature profile closest to the reference simulation was once again the RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model on the polyhedral mesh with 11 prism layers and the AUSM+ inviscid flux discretization scheme.

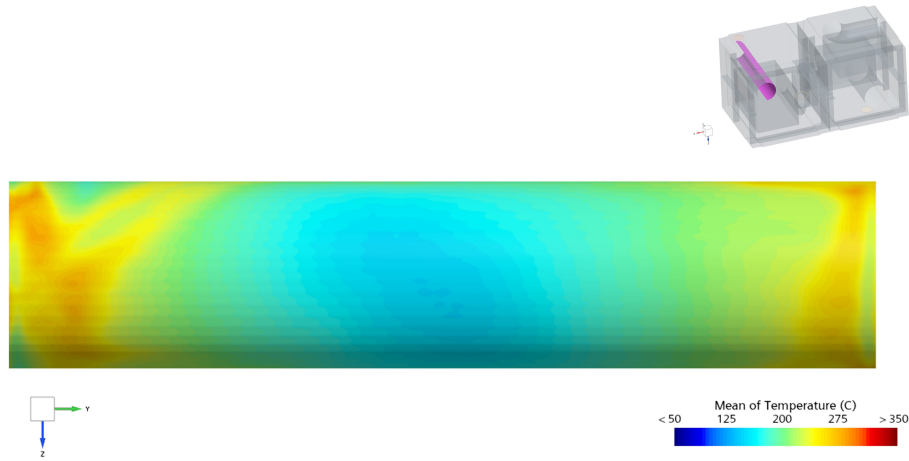


Figure 4.66: Reference simulation showing the temperature of the front surface of the isolated pipe in the aft bay. The location of the surface is shown highlighted in the top right of the figure.



(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers.



(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.67: RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.



(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers.



(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.68: RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.



(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers.



(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the trimmed mesh with 11 prism layers.

Figure 4.69: RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers.



(b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.70: RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the polyhedral mesh with 20 prism layers.



(b) RANS simulation using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme.

Figure 4.71: RANS simulations showing the temperature of the front surface of the isolated pipe in the aft bay.

The reference temperature of the bottom surface of avionics 2 is shown in Figure 4.72. Two low-temperature areas, split by a line of higher temperature, are visible on the surface. The profile is the result of the separation from the avionics pipe shown in Figure 4.20.

Figures 4.73 - 4.77 show the predicted temperature of the RANS simulations on the same surface. As previously discussed, none of the RANS simulations capture the separation behind the avionics pipe. This means that the colder regions predicted by the reference simulation do not appear in any of the results from the RANS simulations. The RANS simulations based on meshes without prism layers,

Figures 4.74a & 4.74b, show temperature profiles where no cold region or hot region is predicted; the temperature is quite even on the entire surface in both simulation results. The RANS simulation using the SST $k-\omega$ turbulence model on the trimmed mesh shown in Figure 4.75a overpredicts the temperature along the forward and aft edges, corresponding to the right and left edges in the image, creating hotspots that do not show up in any of the other simulation results. These three results indicate that the simulation setups without prism layers and the simulation setup using the SST $k-\omega$ turbulence model on a trimmed mesh perform worse for capturing the temperature profile of the surface compared to the results of the other RANS simulations.

The RANS simulations utilizing the Lag EB ε turbulence model on a mesh with prism layers, Figures 4.73b & 4.75b, predict more separation from the avionics pipe than the RANS simulations using the SST $k-\omega$ turbulence model in any form. Two predicted low-temperature zones are shown in the results from the simulations with the Lag EB $k-\varepsilon$ model; however, the aft region appears closer to the forward edge than the aft region of the reference simulation. All RANS simulations with prism layers underpredict the lowest temperature of the cold region and predict a wider spread of the cooling effect towards the left edge, the bottom edge in the figures, than the reference simulation.

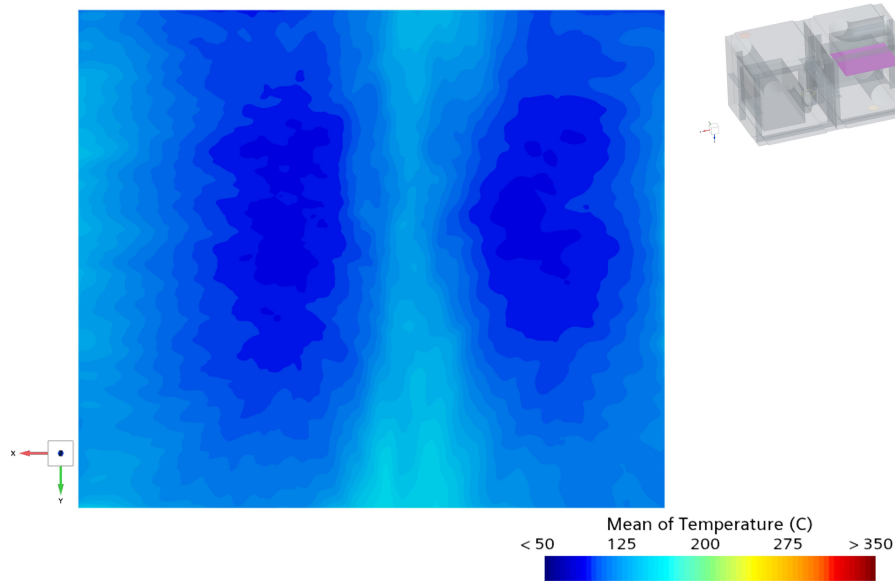
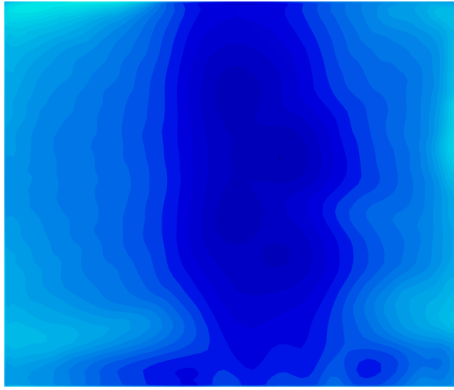
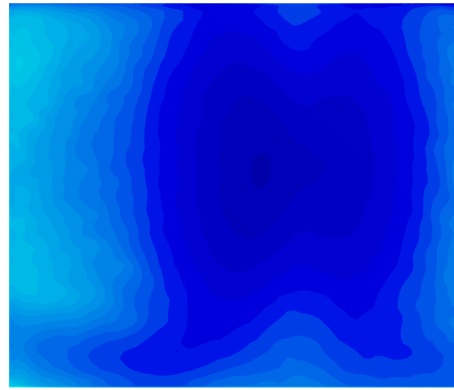


Figure 4.72: Reference simulation showing the temperature of the bottom surface of avionics 2. The location of the surface is shown highlighted in the top right of the figure.

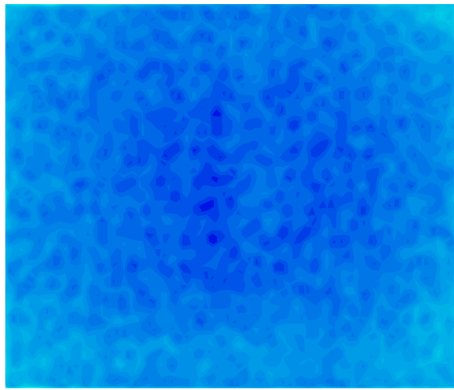


(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh with 11 prism layers.

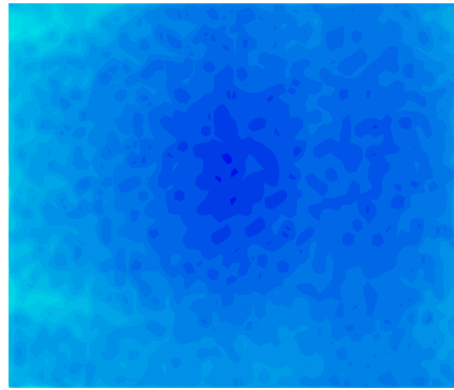


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh with 11 prism layers.

Figure 4.73: RANS simulations showing the temperature of the bottom surface of avionics 2.

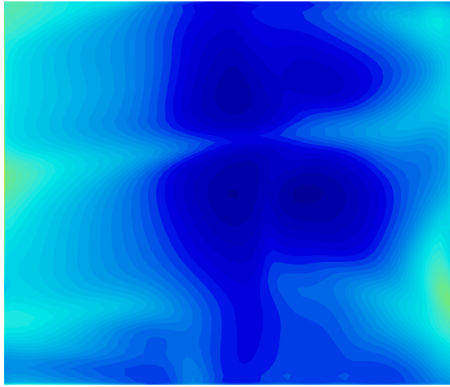


(a) RANS simulation using the SST $k - \omega$ turbulence model and the polyhedral mesh without prism layers.

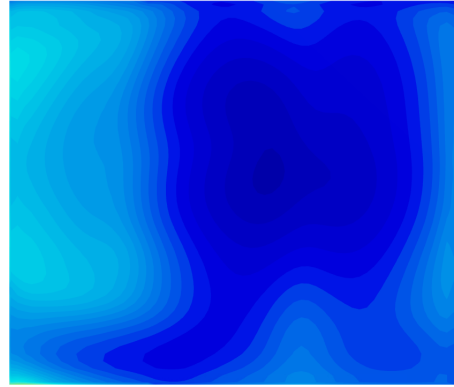


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the polyhedral mesh without prism layers.

Figure 4.74: RANS simulations showing the temperature of the bottom surface of avionics 2.

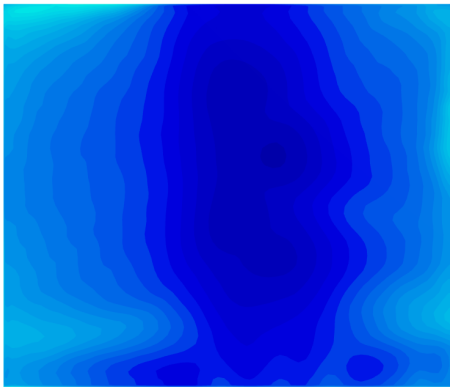


(a) RANS simulation using the SST $k - \omega$ turbulence model and the trimmed mesh with 11 prism layers.

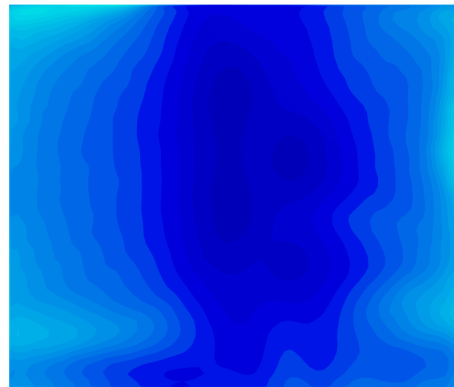


(b) RANS simulation using the Lag EB $k - \varepsilon$ turbulence model and the trimmed mesh with 11 prism layers.

Figure 4.75: RANS simulations showing the temperature of the bottom surface of avionics 2.

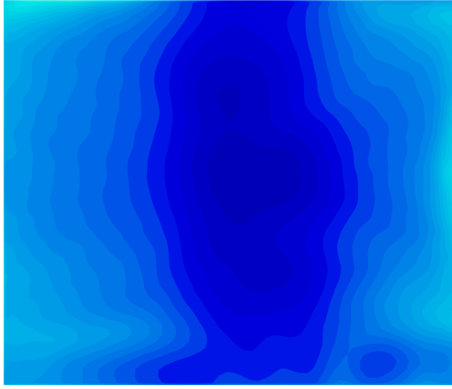


(a) RANS simulation using the SST $k - \omega$ turbulence model with the γ transition model and the polyhedral mesh with 11 prism layers.

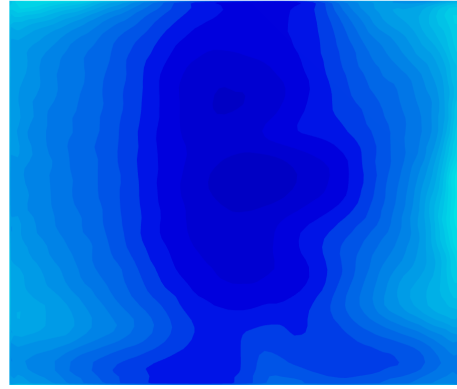


(b) RANS simulation using the SST $k - \omega$ turbulence model with the $\gamma - Re\theta$ transition model and the polyhedral mesh with 11 prism layers.

Figure 4.76: RANS simulations showing the temperature of the bottom surface of avionics 2.



(a) RANS simulation using the SST $k - \omega$ turbulence model with the polyhedral mesh with 20 prism layers.



(b) RANS simulation using the SST $k - \omega$ turbulence model with $\gamma - Re\theta$ transition model, polyhedral mesh with 11 prism layers and the AUSM+ inviscid scheme.

Figure 4.77: RANS simulations showing the temperature of the bottom surface of avionics 2.

4.4 Resource consumption

The resource consumption for each simulation is presented in Table 4.5. The highly resolved simulation using the SRH model was run using 1984 cores, whereas all the RANS simulations were run on 192 cores, less than 10 % of the number used for the SRH simulation. This clearly demonstrates the need for accurate, simplified RANS simulations. There is a significant reduction in simulation time for the simulations using the trimmed mesh compared to the simulations using the polyhedral mesh. This might be due to the trimmed mesh containing roughly 2 million cells fewer than the polyhedral mesh. The SST $k - \omega$ simulations without transition models take a slightly longer time compared to the Lag EB $k - \varepsilon$ model, on the smaller meshes (the polyhedral mesh without prism layers and the trimmed cell mesh) than it does on the larger polyhedral mesh with prism layers. This is interesting as the Lag EB $k - \varepsilon$ turbulence model solves one transport equation more than the SST $k - \omega$ turbulence model without transition models, suggesting that the latter should be faster on all meshes. The simulation with 20 prism layers in the mesh took the longest time to complete at 21.4 hours, while producing similar or worse results to the simulation with the same setup but with only 11 prism layers. An interesting result in the table is that the solution time for the simulation with the AUSM+ inviscid flux discretization scheme was slightly lower, 14.8 h, than the solution time for the same simulation setup with the Roe's inviscid flux discretization scheme. Yet, the simulation with the AUSM+ scheme produced temperature results closer to the predictions of the reference simulation.

Table 4.5: The resource consumption for each simulation.

Simulation	CPU time/iteration	CPU hours	Physical time
Reference	3447 s	744083 h	15.6 days
RANS _{V1}	249 s	2569 h	13.8 h
RANS _{V2}	285 s	2695 h	14.0 h
RANS _{V3}	124 s	996 h	5.2 h
RANS _{V4}	111 s	921 h	4.8 h
RANS _{V5}	162 s	1354 h	7.1 h
RANS _{V6}	161 s	1235 h	6.4 h
RANS _{V7}	303 s	2738 h	14.4 h
RANS _{V8}	354 s	2870 h	15.1 h
RANS _{V9}	515 s	4081 h	21.4 h
RANS _{V10}	277 s	2829 h	14.8 h

5

Conclusion

In general, the RANS simulation that had the best accuracy for predicting the temperature profile of the walls was the simulation using the SST $k-\omega$ turbulence model with the $\gamma-Re\theta$ transition model on the polyhedral mesh with 11 prism layers with the AUSM+ inviscid flux discretization scheme. This RANS simulation also showed the least amount of errors when predicting the surface-averaged mean temperature of the avionics and pipes inside the internal bay. The Lag EB $k-\varepsilon$ model lacked accuracy in several regions, as well as in the surface- and volume-averaged temperatures, when compared to the SST $k-\omega$ model on the polyhedral mesh. The Lag EB $k-\varepsilon$ model was, however, more accurate on the trimmed mesh than the SST $k-\omega$ model was in both the surface- and volume-averaged temperatures and heat transfer.

Comparing the simulations with prism layers to the simulations without prism layers, a clear difference is visible in the right cross-sectional plane. The simulations without prism layers in the mesh were severely overpredicting the temperature in the internal bay by around 50 °C to 100 °C. Prism layers were required for capturing the heat transfer and temperature of the boundaries inside the bay accurately, as was shown in Section 4.3.2. The simulations using the polyhedral mesh with 11 prism layers showed results closer to the reference case than the simulations using the trimmed mesh with 11 prism layers, both for the surface- and volume-averaged temperatures and heat transfer, and in many of the temperature and velocity profiles. The simulation based on the polyhedral mesh with 20 prism layers showed no real improvement compared to the simulation based on the polyhedral mesh with only 11 prism layers.

The transition models for the SST $k-\omega$ turbulence model showed great potential, with the $\gamma-Re\theta$ model predicting values and profiles close to the reference simulation in many cases. Switching from Roe's inviscid flux discretization scheme to the AUSM+ inviscid flux discretization scheme showed an increased performance in capturing flow characteristics and surface temperatures. The AUSM+ scheme was also cheaper in terms of the required solving time for a set number of iterations.

Regarding the sustainability aspect of this work, a reduction in simulation time means a reduction in resource consumption. This reduces the power consumption per simulation and the need for large data centers, which is beneficial for the environment.

For simulations of similar character, the RANS simulation using the SST $k-\omega$ turbulence model with the quadratic constitutive relationship and the $\gamma - Re\theta$ transition model run on a polyhedral mesh with prism layers using the AUSM+ inviscid flux discretization scheme will provide results within roughly 4 % of the time-averaged results of a highly resolved turbulent simulation, for a much shorter simulation time, as shown in this thesis.

5.1 Future work

Future work would be to evaluate different prism layer setups to find a general setup that captures the thermal boundary layer accurately. An optimization study comparing the measured temperature and heat transfer of the boundaries for different placements of inlet, outlet and ventilation hole/holes might provide general guidelines for designing an internal bay. A CHT on simulation complete with thermal radiation of an internal bay would also be recommended, as the heat transfer in the wall separating the two bays would most likely have a large impact on the fluid temperature in the front bay.

A downside of the SRH simulation is that the flow field near the wall is computed using RANS. For temperature and heat transfer simulations, the thermal boundary layer close to the walls is important. Using RANS near the walls introduces errors, which might have a significant impact on the results shown in this thesis. For a more accurate study, a LES or Direct Numerical Simulation (DNS) would be a better reference if experimental data is not viable.

This thesis has not studied the convergence rates of the different RANS simulations. Such a study might be interesting to determine which RANS simulation setup finds the most accurate solution in the least number of iterations, further reducing the computational cost.

A numerical study of how different turbulence models are affected by different inviscid flux discretization schemes would also be interesting, as the AUSM+ inviscid flux discretization scheme showed great potential with the SST $k - \omega$ turbulence model in this thesis.

Bibliography

- [1] M. Geron, C. Butler, J. Stafford, and D. Newport, “Development and validation of a compact thermal model for an aircraft compartment,” *Applied Thermal Engineering*, vol. 61, no. 2, pp. 65–74, 2013, ISSN: 1359-4311. DOI: <https://doi.org/10.1016/j.applthermaleng.2013.07.012>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1359431113005000>.
- [2] A. Akin and H. S. Kahveci, “An optimization study for rotorcraft avionics bay cooling,” *Aerospace Science and Technology*, vol. 90, pp. 1–11, 2019, ISSN: 1270-9638. DOI: <https://doi.org/10.1016/j.ast.2019.04.029>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1270963819305267>.
- [3] C. Butler and D. Newport, “Experimental and numerical analysis of thermally dissipating equipment in an aircraft confined compartment,” *Applied Thermal Engineering*, vol. 73, no. 1, pp. 869–878, 2014, ISSN: 1359-4311. DOI: <https://doi.org/10.1016/j.applthermaleng.2014.08.035>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S1359431114006978>.
- [4] L. Davidson. “Tensors,” Accessed: May 15, 2026. [Online]. Available: https://www.cfd-sweden.se/lada/comp_fluid_dynamics/postscript_files/chapter_tensorer.pdf.
- [5] L. Davidsson. “Fluid mechanics, turbulent flow and turbulence modeling,” Accessed: Jun. 27, 2026. [Online]. Available: https://www.cfd-sweden.se/lada/postscript_files/solids-and-fluids_turbulent-flow_turbulence-modelling.pdf.
- [6] H. V. Malalasekera and W., *An introduction to computational fluid dynamics: the finite volume method*. Pearson Prentice Hall, 2007.
- [7] “Star-CCM+ Documentation,” Accessed: Jun. 27, 2026.
- [8] V. Duffal, B. de Laage de Meux, and R. Manceau, Eds., *Development and Validation of a Hybrid RANS-LES Approach Based on Temporal Filtering*, vol. 2: Computational Fluid Dynamics, Fluids Engineering Division Summer Meeting, Jul. 2019, V002T02A053. DOI: 10.1115/AJKFluids2019-4937. eprint: <https://asmedigitalcollection.asme.org/FEDSM/proceedings-pdf/AJKFluids2019/59032/V002T02A053/6449370/v002t02a053-ajkfluids2019-4937.pdf>. [Online]. Available: <https://doi.org/10.1115/AJKFluids2019-4937>.

- [9] “Simscale documentation,” Accessed: Apr. 16, 2026. [Online]. Available: <https://www.simscale.com/docs/simulation-setup/global-settings/k-omega-sst/>.
- [10] F. Menter, M. Kuntz, and R. Langtry, “Ten years of industrial experience with the sst turbulence model,” *Heat and Mass Transfer*, vol. 4, Jan. 2003.
- [11] P. R. Spalart and C. L. Rumsey, “Effective inflow conditions for turbulence models in aerodynamic calculations,” *AIAA Journal*, vol. 45, no. 10, pp. 2544–2553, 2007. DOI: 10.2514/1.29373. eprint: <https://doi.org/10.2514/1.29373>. [Online]. Available: <https://doi.org/10.2514/1.29373>.
- [12] P. Durbin, “On the k-3 stagnation point anomaly,” *International Journal of Heat and Fluid Flow*, vol. 17, no. 1, pp. 89–90, 1996, ISSN: 0142-727X. DOI: [https://doi.org/10.1016/0142-727X\(95\)00073-Y](https://doi.org/10.1016/0142-727X(95)00073-Y). [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0142727X9500073Y>.
- [13] S. Lardeau, “Development of an elliptic-blending lag model for industrial applications,” *54th AIAA Aerospace Sciences Meeting*, Jan. 2016. DOI: 10.2514/6.2016-1600.
- [14] R. Manceau and K. Hanjalić, “Elliptic blending model: A new near-wall reynolds-stress turbulence closure,” *Physics of Fluids*, vol. 14, no. 2, pp. 744–754, Feb. 2002, ISSN: 1070-6631. DOI: 10.1063/1.1432693. eprint: https://pubs.aip.org/aip/pof/article-pdf/14/2/744/19267401/744_1_online.pdf. [Online]. Available: <https://doi.org/10.1063/1.1432693>.
- [15] O. Wiström and W. Wallace, “The effects of turbulence modelling in cfd for heat transfer in aircraft,” M.S. thesis, Linköpings Universitet, 2023.
- [16] M. Sosnowski, J. Krzywanski, K. Grabowska, and R. Gnatowska, “Polyhedral meshing in numerical analysis of conjugate heat transfer,” in *European Physical Journal Web of Conferences*, ser. European Physical Journal Web of Conferences, vol. 180, EDP, Nov. 2018, 02096, p. 02096. DOI: 10.1051/epjconf/201818002096.
- [17] J. W. Slater, Ed. “Examining spatial (grid) convergence,” Accessed: Feb. 25, 2026. [Online]. Available: <https://www.grc.nasa.gov/www/wind/valid/tutorial/spatconv.html>.
- [18] I. B. Celik, U. Ghia, P. J. Roache, C. J. Freitas, H. Coleman, and P. E. Raad, “Procedure for estimation and reporting of uncertainty due to discretization in cfd applications,” *Journal of Fluids Engineering*, vol. 130, no. 7, p. 078001, Jul. 2008, ISSN: 0098-2202. DOI: 10.1115/1.2960953. eprint: https://asmedigitalcollection.asme.org/fluidsengineering/article-pdf/130/7/078001/5491455/078001_1.pdf. [Online]. Available: <https://doi.org/10.1115/1.2960953>.
- [19] L. Davidson, “Large eddy simulations: How to evaluate resolution,” *International Journal of Heat and Fluid Flow*, vol. 30, no. 5, pp. 1016–1025, 2009, The 3rd International Conference on Heat Transfer and Fluid Flow in Microscale, ISSN: 0142-727X. DOI: <https://doi.org/10.1016/j.ijheatfluidflow.2009.06.006>. [Online]. Available: <https://www.sciencedirect.com/science/article/pii/S0142727X09001039>.

- [20] “Two equation gamma theta transition model,” Accessed: Jun. 27, 2026. [Online]. Available: https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v251/en/cfx_thry/i168504023837015.html.
- [21] C. Johansson. “Transition modelling in simcenter star-ccm+,” Accessed: Jun. 27, 2026. [Online]. Available: <https://volume.com/simcenter-star-ccm/transition-modelling-in-simcenter-star-ccm/>.
- [22] “Navigating cfd flow solvers: Segregated vs. coupled methods their applications,” Accessed: May 27, 2026. [Online]. Available: <https://www.resolvedanalytics.com/cfd/coupled-vs-segregated-cfd-flow-solvers>.
- [23] P. Ekman, *Important Factors for Accurate Scale-Resolving Simulations of Automotive Aerodynamics*. May 2020, ISBN: 9789179298630. DOI: 10.3384/diss.diva-164926.
- [24] J. Murad. “What is y^+ (yplus)?” Accessed: Jun. 27, 2026. [Online]. Available: <https://www.simscale.com/forum/t/what-is-y-yplus/82394>.
- [25] L. Davidson. “How to estimate the resolution of an les of recirculating flow,” Accessed: Jun. 27, 2026. [Online]. Available: https://www.cfd-sweden.se/lada/postscript_files/paper_qles2009.pdf.



CHALMERS
UNIVERSITY OF TECHNOLOGY