



**University
of Manitoba**



Letter of Transmittal

December 7, 2022

Elizabeth Dafoe Library
University of Manitoba.
25 Chancellors Circle,
Winnipeg, MB R3T 2N2

Dear Instructional Team,

This report was prepared by a team comprised of students undertaking the MECH 4860 - Engineering Design Course offered at the University of Manitoba for Fall 2022. New Flyer Industries is the client who provided the team with the engineering challenge that is explored within the contents of this report. This report contains the work done by the students to meet the project objectives and produce the required deliverables as established by the client.

The focus of the report is to provide a design concept to the client aimed at substantiating the application of an air curtain to reduce heat loss through the door in an electric bus. In this report, a brief background on New Flyer and the challenge they presented us with is provided. The report then outlines an overview of how the problem was tackled and the methodology used to do so. The main portion of the report is centered around the design and analysis of an air curtain to reduce heat losses through the bus door during passenger ingress and egress. The majority of the analysis was completed using STAR-CCM+ to conduct computational fluid dynamic studies. The report then shares future recommendations identified by the team to be used for further analysis and implementation of the air curtain design. Finally, the report ends with a conclusion that summarizes the design process and the final results obtained for the air curtain design.

The team would like to acknowledge New Flyer for their support of this project, specifically Paul Cantin and Jason Newton who were integral in providing guidance. We would also like to thank Dr. Qingjin Peng for his support and technical engineering expertise, and Aidan Topping for sharing her knowledge on technical communication which elevated the quality of this report.

Sincerely,

Team 18
Jack Zonneveld
Deepak Bhudia
Ashir Ahmed
Stephen Crew



**University
of Manitoba**



New Flyer Industries Bus Door Air Curtain Design and Analysis

PRICE FACULTY OF ENGINEERING
MECH 4860 - ENGINEERING DESIGN

Report Prepared For: Dr. Paul Labossiere
Paul Cantin
Dr. Qingjin Peng
Aidan Topping

Report Prepared By: Jack Zonneveld
Deepak Bhudia
Ashir Ahmed
Stephen Crew

Date of Submission: Wednesday, December 7th, 2022

Executive Summary

This project explores the design and analysis of an air curtain to be used in New Flyer's electric bus. New Flyer is a leading bus manufacturer in North America and is working towards increasing the efficiency of the Xcelsior Charge line of electric buses. New Flyer identified the door as a major energy loss source during passenger ingress and egress. An air curtain is proposed in the report to reduce energy loss through the door. The proposed design forms an air boundary separating the outside atmosphere from the bus cabin atmosphere to reduce heat transfer through the door boundary when the doors are open.

The team had the task to develop a solution to reduce heat transfer through the door. The design is required to meet industry standards outlined by American Public Transport Agency (APTA), and the Americans with Disabilities Act (ADA). Although no specific project budget was set, cost was minimized where possible and influenced design decisions. The final design was to be submitted to the client as a CAD model along with the results of the analysis obtained. The submission to the client also included Failure Modes and Effects Analysis (FMEA) for designing, manufacturing, and installation of the air curtain. In addition, future recommendations concerning both the design and analysis are included in the report.

STAR-CCM+ was utilized to perform a Computational Fluid Dynamic (CFD) analysis of the air curtain design. Initial analysis of the air curtain did not show promising results and hence three more studies were performed to explore the air curtain performance. Simulation data showed increased heat transfer rates for the door equipped with the air curtain, however, it was demonstrated that the air curtain was successful in creating a barrier to separate internal and external air. Due to the lack of HVAC return in the simplified CFD model, the air leaving the air curtain plenum was exiting the bus at an increasing rate. With optimization of the HVAC return, nozzle position, angle, and cross-sectional area, the observed rate of heat transfer through the door would improve.

This report includes the steps taken to design and perform CFD analysis on the air curtain, along with the challenges encountered by the team. Additionally, base case CFD analysis (without the air curtain) was performed to compare the results with the air curtain analysis to help understand the value added.

The final design proposed to the client is a dual fan-operated air curtain. Centrifugal fans used in the air curtain are operated using 24 VDC with a power rating of 175.2 W each to produce 2200 CFM combined. A distinct feature of the design is its compact geometry which takes up less space and requires fewer changes to the base plate. The geometry design quickly spreads the flow along the length of the door as the fans are activated to form an air door when the bus door is opened. High-density polyethylene (HDPE) used for the design makes it highly impact resistant allowing it to meet the durability requirements for the transit bus application. The energy consumption of the fans used is 0.741 kWh for one day of operation which is significantly lower than the average energy consumption by the HVAC system used to heat and cool the bus with an average daily consumption of 70.57 kWh in Winnipeg.

The total cost for producing a prototype for testing is approximately \$4480, which includes the adjustments to the CAD modeling, rework instruction for the prototype air curtain, rapid prototyping, and installation covering both mechanical and electrical work involved. The cost rises up to \$16000 for a detailed production-ready design for implementation in the bus.

Table of Contents

List of Figures	v
List of Tables	vi
1 Introduction	1
1.1 Problem Statement and Project Objective	1
1.2 Limitations and Design Constraints	1
1.3 Industry Specifications	2
1.4 Technical Specifications and Client Needs	2
1.4.1 Product Needs	2
1.4.2 House Of Quality	3
1.5 Project Deliverables	4
1.6 Project Changes	5
1.6.1 Deadspace Closeout Elimination	5
1.6.2 Closed Door Sealing Elimination	6
2 Methodology	6
2.1 CAD Modeling/Pre-Processing	6
2.2 CFD Methodology	7
2.2.1 CFD Software	7
2.2.2 Set-up & Boundary Conditions	7
2.2.3 Mesh	9
2.2.4 Post-Processing	10
3 Concept Development, Analysis and Selection	10
3.1 Plenum Designs	10
3.1.1 Design #1	10
3.1.2 Design #2	11
3.1.3 Design #3	11
3.1.4 Design #4	13
3.2 Concept Screening	13
3.3 Fan Selection	15
3.3.1 Axial Fan	15
3.3.2 Tangential Fan	15
3.3.3 Centrifugal Fan	15
3.4 Exit Velocity	16
4 Analysis	16
4.1 Energy Consumption Calculations	17
4.2 Heat Transfer Through the Door	18
4.2.1 Closed Door	18
4.2.2 Open Door	19
4.3 Base Case CFD Analysis	21
4.3.1 CFD Parameters	21
4.3.2 Velocity Streamlines	22
4.3.3 Temperature Gradient	22
4.3.4 Heat Transfer Coefficient	23
4.3.5 Boundary Heat Flux	23
4.3.6 Mass Flow Rate Through Door Boundary	24
4.3.7 Heat Transfer Through Door Boundary	24
4.3.8 Wall Y+ Histogram	25
4.3.9 Convective Courant Number Histogram	25
4.3.10 Residuals	26

4.4	Optimized Base Case CFD Analysis	27
4.4.1	CFD Parameters	28
4.4.2	Velocity Streamlines	28
4.4.3	Temperature Gradient	29
4.4.4	Heat Transfer Coefficient	29
4.4.5	Boundary Heat Flux	30
4.4.6	Mass Flow Rate Through Door Boundary	30
4.4.7	Heat Transfer Through Door Boundary	31
4.4.8	Wall Y+ Histogram	31
4.4.9	Convective Courant Number Histogram	31
4.4.10	Residuals	32
4.5	Initial Analysis of Air Curtain	32
5	Final Design	33
5.1	Air Curtain CFD Analysis	34
5.1.1	CFD Parameters	34
5.1.2	Velocity Streamlines	35
5.1.3	Temperature Gradient	35
5.1.4	Heat Transfer Coefficient	36
5.1.5	Boundary Heat Flux	37
5.1.6	Air Mixing	37
5.1.7	Mass Flow Rate Through Door Boundary	38
5.1.8	Heat Transfer Through Door Boundary	39
5.1.9	Wall Y+ Histogram	39
5.1.10	Convective Courant Number Histogram	40
5.1.11	Residuals	40
5.2	Fan	41
5.3	Control System	43
5.4	Cost Analysis	44
5.5	Failure Modes & Effects Analysis (FMEA)	45
6	Future Recommendations	46
6.1	Future Air Curtain Analysis	47
6.2	Future Air Curtain Design Considerations	49
6.3	Additional Energy Saving Designs	51
7	Conclusion	53
8	References	54
A	Centrifugal Fan Specifications	56
B	manufacturing Quote	57
B.1	Design manufacturing quote	57
B.2	Roto-molding charges	58
C	FMEA Rubrics	58
C.1	Severity of Effect	58
C.2	Failure Probability	59
C.3	Likelihood of Risk Detection	59
D	Engineering Drawing	60

List of Figures

1	Dead space areas highlighted on entrance door.	5
2	CFD base case model.	6
3	External volume subtract for CFD model, showing the passenger door interface.	7
4	Base case - split by patch operation to split surfaces of interest and create boundaries/interfaces.	8
5	Base case - regions and boundary creation.	9
6	Surface mesh for (a) external and internal fluid regions and (b) conformal door mesh.	10
7	Design #1 renders.	11
8	Design #2 renders.	11
9	Design #3 - plenum top view in passenger door assembly.	12
10	Design #3 - packaging of plenum with existing passenger door assembly.	12
11	Design #3 renders.	13
12	Off-the-shelf air curtain.	13
13	Tangential fan.	15
14	Centrifugal fan.	16
15	Flow rate relation to enclosure volume and opening depth.	20
16	Flow rate relation with wind velocity and opening size.	20
17	Base case - velocity streamlines through door boundary.	22
18	Base case - temperature gradients through door boundary and propagation into external air medium for (a) 1 s and (b) 7 s of transient solution.	22
19	Base case - heat transfer coefficient through door boundary.	23
20	Base case - boundary heat flux through door boundary.	24
21	Base case - transient mass flow rate through door boundary.	24
22	Base case - transient heat transfer response through door boundary.	25
23	Base case - wall Y+ histogram.	25
24	Base case - convective Courant number histogram.	26
25	Base case - residuals.	26
26	CFD CAD model comparison - fluid volume region for (a) base case and (b) optimized base case.	27
27	CFD mesh comparison - volume mesh for (a) base case and (b) optimized base case.	28
28	Base case - velocity streamlines through door boundary and propagation into external air medium for (a) 0 s and (b) 7 s of transient solution.	28
29	Optimized base case - temperature gradients through door boundary and propagation into external air medium for (a) 0 s and (b) 7 s of transient solution.	29
30	Optimized base case - heat transfer coefficient through door boundary for (a) 0 s and (b) 7 s of transient solution.	29
31	Optimized base case - boundary heat flux through door boundary.	30
32	Optimized base case - transient mass flow rate through door boundary.	30
33	Optimized base case - transient heat transfer response through door boundary.	31
34	Optimized base case - wall Y+ histogram.	31
35	Optimized base case - convective Courant number histogram.	32
36	Optimized base case - residuals.	32
37	Initial analysis for 860 CFM.	33
38	Final Design - air curtain with entrance door in open position.	33
39	Final Design - air curtain (a) front view and (b) side view in context of the bus A-class surface.	34
40	Air curtain - velocity streamlines through door boundary and propagation into external air medium for (a) 1 s and (b) 7 s of transient solution.	35
41	Air curtain - temperature gradients through door boundary and propagation into external air medium for (a) 1 s and (b) 7 s of transient solution.	36
42	Air curtain - heat transfer coefficient through door boundary.	36
43	Air curtain - boundary heat flux through door boundary.	37
44	Air curtain - air mixing throughout a 7 s door cycle.	38
45	Air curtain - transient mass flow rate through door boundary.	39

46	Air curtain - transient heat transfer response through door boundary.	39
47	Air curtain - wall Y^+ histogram.	40
48	Air curtain - convective Courant number (CFL) histogram.	40
49	Air curtain - residuals.	41
50	Centrifugal fan for the final design.	41
51	Magnetic and mechanical door contact system.	44
52	Predicted affected airflow region during passenger ingress and egress.	47
53	Location of curbside front defrost air.	49
54	Entrance door top panel (blue) and top baseplate (red) isometric view.	50
55	Entrance door top panel new shape.	50
56	Cut-out region for plenum nozzle in entrance door top panel.	51
57	Dynamic Pressure Analysis of a Bus [20].	52
A.1	Specification for centrifugal fan.	56
B.1	Price quote from the manufacturing.	57
B.2	Plum Roto-molding charges.	58
D.1	Air curtain plenum drawing.	60

List of Tables

I	HOUSE OF QUALITY	4
II	CFD SOLVER PARAMETERS	8
III	POLYHEDRAL MESH PARAMETERS USED DURING CFD SIMULATIONS	9
IV	PLENUM SELECTION METRIC	13
V	SUMMARIZED DAILY HVAC LOADS FOR NFI FLEET	16
VI	ADJUSTED BATTERY CAPACITY AND EFFICIENCY OF NEW FLYER ELECTRIC BUSES	17
VII	HVAC ENERGY CONSUMPTION IN NEW FLYER BUSES	18
VIII	CLOSED DOOR HEAT TRANSFER SUMMARY	19
IX	OPEN DOOR HEAT TRANSFER SUMMARY	20
X	BASE CASE - CFD SIMULATION INPUT PARAMETERS AND BOUNDARY CONDITIONS	21
XI	OPTIMIZED BASE CASE - CFD SIMULATION INPUT PARAMETERS AND BOUNDARY CONDI- TIONS	28
XII	AIR CURTAIN - CFD SIMULATION INPUT PARAMETERS AND BOUNDARY CONDITIONS . . .	34
XIII	FAN ENERGY USE SUMMARY FOR 12-HOUR OPERATION	43
XIV	ROTO-MOLDING CHARGES	44
XV	COST FOR PROTOTYPE DESIGN	45
XVI	COST FOR DETAILED PRODUCTION	45
XVII	FMEA OF AIR CURTAIN	45
XVIII	IRPN OF AIR CURTAIN	46
XIX	SEVERITY OF EFFECT	58
XX	FAILURE PROBABILITY	59
XXI	LIKELIHOOD OF RISK DETECTION	59

1 Introduction

Everyday millions of people rely on public transit for their daily commuting needs. As cities continue to grow the demand for improved infrastructure and technology centered around the public transit industry is rapidly increasing. For most cities buses are the focal point of their public transport systems, which has created a huge demand for the best buses that the industry has to offer. Cities continue to invest billions of dollars each year into improved public transportation systems as they offer an opportunity to improve the day-to-day lives of citizens. Such improvements stem from the fundamental idea that a bus can transport several vehicles worth of people while requiring less space on the roads and for parking which leads to improved traffic flow. For the same reasons, buses can provide transportation at a much cheaper energy cost per person which in the setting of the modern world helps to address the issue of climate change.

The challenge of dealing with climate change and reducing the emissions produced by the transportation industry is also being addressed through the transition from gas and diesel fueled vehicles to electric vehicles. This massive shift in the industry is also extremely relevant for buses and has led to a race among bus producers worldwide to provide customers with top of the line electric buses.

Currently, New Flyer Industries (NFI) based out of Winnipeg, Manitoba is the industry leader in North America for both the design and production of public transit buses. NFI has approximately 7,500 employees working in 9 countries [1]. New Flyer supports over 35,000 buses currently in service, around 8,600 of them being propelled using electric motors and on-board batteries, and 1,900 being zero emission [1]. As the current industry leader for the North American bus and coach market, New Flyer is constantly challenging themselves to improve the products they have to offer their customers.

In electric buses, such as the Xcelsior Charge line of buses produced by NFI, a major challenge is improving and optimizing the range of the buses. NFI has identified the current heating, ventilation, and air conditioning (HVAC) system as a major source of lost energy from the battery as the current system relies solely on energy provided by the battery. This is a new challenge since diesel buses are capable of recycling heat from the inefficient combustion process for reuse in the cabin. With this new challenge at hand, New Flyer identified energy loss through the door as a key contributor to wasted battery power and has challenged the team to come up with a design to address this issue.

To develop a solution to this problem, the team has gone through a detailed design process that involved clearly defining the problem and developing concepts based on the key elements of the problem. This report will outline these phases of the project and then provide a detailed design and corresponding analysis that is to be presented to the client.

1.1 Problem Statement and Project Objective

NFI's current goal is to introduce electrical buses in cities all over the world. NFI has identified the problem of air leakage through its door system, during passenger ingress and egress, as a large factor contributing to an increased HVAC energy demand which reduces the range of the bus. NFI has entrusted the team to propose a solution which reduces the air exchange between the cabin and ambient air.

1.2 Limitations and Design Constraints

On September 19, 2022, the team had a discussion with NFI during an on site visit to review the needs and constraints of the project. The constraints were based on the industry specification set out by the Americans with Disabilities Act (ADA), and the American Public Transport Agency (APTA). Further constraints were added by the client based on design and production requirements.

After meeting with the client to clarify expectations, they highlighted the importance of a design to have minimal downstream impacts on existing bus systems. These critical systems include HVAC, body structure, and architectural changes to the bus door base-plate envelope. New Flyer requested that the priority be given to the concept that integrates seamlessly with existing bus infrastructure in order to reduce capital expenditure on retooling. A system that integrates into the existing door system also has the advantage of being backwards compatible with older buses that are already in service.

Time is an important constraint in this project due to the open-ended scope and computationally intensive CFD workflow. Thus, the design process must be carefully planned out and the scope limited in a manner that allows for completion of the project in the allotted time.

1.3 Industry Specifications

To successfully design a product that meets the client's needs, the team has to design a product that follows relevant ADA and APTA specifications. The following specifications need to be satisfied by the new design.

- The new design needs to preserve the cabin temperature within the given range as described by APTA HVAC temperature set point in TS 55 [2]. For the cool weather application that will be used for analysis, this set temperature is 15°C.
- The door must still conform to TS 80 which states that the door must have a minimum possible opening width of $31\frac{3}{4}$ inches [2]. This specification also outlines the push out force requirements for the door.
- As outlined in TS 56.1, passengers must not be exposed to airflow faster than 100 ft/min from the HVAC system [2].
- The use of any mechanical equipment in the design must have a sound level below 70 dBA as specified in TS 5.8.1 [2]. This is to reduce the sound level exposure on the passengers.
- As per TS 56.2, the addition of any new product to the bus must ensure that the windshield defrost system meets SAE J382 specifications. This SAE specification is put in place to guarantee adequate visibility for the driver.

1.4 Technical Specifications and Client Needs

Initial technical specifications and needs for the client were identified during the project definition phase. To get a further understanding of the client's needs, meetings were held with the client that expanded the team's understanding of the project requirements. The following section outlines the client's needs and specifications required that must be met. The final technical specifications and client needs are summarized in Table I.

1.4.1 Product Needs

The following list of needs were identified by categorizing the exhaustive list of needs developed during the project definition phase. The condensed list is presented below.

1. The new product design should increase the range of the bus. This is of major importance to the project as it is directly related to the project objective.
2. The design is cost and weight efficient compared to current design.
3. The design should be easy to use and operate.
4. The new design is easily serviceable and is able to withstand any external factors such as a person leaning on the door.

5. The product design should maintain the function of other bus systems. This includes changes to existing system such as HVAC, electrical, and the body structure of the bus.
6. The product design is safe for passengers and does not compromise safety for efficiency.

The house of quality shown in the following section outlines all the needs identified by the team and their respective weighting against engineering matrices.

1.4.2 House Of Quality

The House of Quality connects the client needs to measurable engineering metrics and assigns a quantifiable relationship between the two. This is used to determine the primary targets for the new design. The importance of each need and metric is given on a scale of one to five, with five being the highest importance and one being of less importance. Table I shows the house of quality.

Table I: HOUSE OF QUALITY

	Engineering Metrics													
	Relative Importance	New door/system reduces energy losses	Total Mass	Added Cost	Door cycle time	System Reponsiveness	Parts last duration of buses life cycle	Drag Force	Visibility	Passes safety tests	Passengers feel comfortable	Door looks good	Noise Level	Installation Time
Client Needs:														
Reduces heat exchange through conductive heat transfer	2	3												
Reduces heat exchange from air leaking when door is closed	4	3					1							
Reduces heat exchange through airflow while door is open	5	3												
Reduces heat exchange through unused space when door is open	4	3							2					
Net energy equation of additional systems is positive	5	3												
New door is the same weight or lighter	2		3		1									
Any new system is lightweight	3		3											
Changes to existing parts are cost effective	4			3										
Additional systems are in-expensive	3			3										
Opens and closes promptly	5	2			3	2								
Opens and closes when its supposed to	5	2				3					1			
Additional system activates when required.	5					3								
Accessible for people with disabilities	5									3	3			
Parts are durable	4						3							
Parts work in extreme climates	3				1		2			1				
Parts work after many cycles	4				1		3							
Parts are easy to access for repair	2													3
Parts are easily be repaired	3													3
Exterior features maintain aerodynamics	1	1						3						
Front window defrost system works	4	1							3	3				
Driver sightlines through the door are clear	5								3	3				
Door is safe to pass through	5									3	1			
Door is safe to operate	5									3	1			
Door has a failsafe mode	4									3	1			
Additional systems are safe	5									3				
Has room to pass through	5										3	2		
Contact free when passing through	3										3	1		
Handles are ergonomic	1										3	1		
Comfortable to lean on	2										3			
Aesthetically pleasing	3								1			3		
Components hidden in empty/closed space	3											3		
Door/systems are quiet	1										3		3	
Easy to install	3													3
Importance	X	85	15	21	24	40	33	3	38	102	70	32	3	24
Ideal Specifications Values	X	>20% Improvement	<50 added	<1000 added	7	0.25	20	existing	= existing'	Pass	Subjective	Subjective	5 added	<5% cell gain
Acceptable Specifications Values	X	>10% Improvement	<100 added	<500 added	7	0.5	15	existing	= existing'	Pass	Subjective	Subjective	10 added	<10% cell gain
Target Specification Units	X	J	lbs	\$	sec	sec	yrs	N	deg	Y/N	Y/N	Y/N	dB	sec

1.5 Project Deliverables

The team will deliver the following to New Flyer post-completion of this project:

- Full CAD model.

- Engineering Bill of Materials (BOM).
- Preliminary Engineering Drawings.
- Post Processed CFD simulation results.
- Presentation highlighting key performance metrics of new design compared to previous door system.
- Energy exchange calculations.
- Failure Modes and Effects Analysis (FMEA).

1.6 Project Changes

The concept generation phase of the project concluded with three design ideas still in consideration. The previously generated ideas of closing the deadspace in the door while in the opened position and improving the seal of the door while in the closed position were eliminated from consideration for the final design. Although these areas won't be addressed in detail within this report they will still be discussed in the future recommendations of the project found in Section 6 as they are still valid ways of improving the overarching problem of reducing energy loss.

The reason for eliminating these from consideration is that a thorough analysis on multiple design ideas could not be completed within the time available to complete the project. Ultimately, the air curtain was chosen as it has more potential for a larger return on the key target specification of reducing energy loss through the door than closing out deadspace or improving the seals did. The following sections will outline the justification for this decision.

1.6.1 Deadspace Closeout Elimination

The deadspace in the door is defined as the area of the door that is unavailable for practical use when the entrance door is in the open position. A graphic that outlines this region is shown in Figure 1.



Figure 1: Dead space areas highlighted on entrance door.

In this figure only a small part of the total open area of the door is encompassed in the deadspace region. Using the base case analysis in Figure 17 it can be seen that the heat transfer through the door is the highest through the middle of the door. Along the perimeter the heat flux is lower thus there will be less energy transferred through the deadspaces in the door. Through combining these ideas, it was concluded that designing for this area would harvest a comparatively lower energy return which is the key target specification for this project.

1.6.2 Closed Door Sealing Elimination

Another design area that was previously being considered was to improve the seal of the door when it is closed. To properly make any design recommendations on this issue the team would need to complete an extensive analysis using lean six sigma principles. This would include compiling a dataset using buses that have been on the roads for years, focusing on the degradation of the door mechanism, rubber seals, and draft brushes. Given the limited time available to complete the project obtaining these statistics is not possible thus this method of addressing heat loss through the door was eliminated from consideration. Another reason for eliminating this idea is that any problems with the seal of the bus would result in very small leaks where the energy loss through these areas would be small compared to the losses through an open door.

2 Methodology

The methodology used throughout the design and numerical analysis of the base case and air curtain simulations is developed below. This includes the CAD pre-processing, CFD methodology, software, and solvers used during the simulations. Further development of the CFD results is presented in Section 4.

2.1 CAD Modeling/Pre-Processing

To accurately characterize the heat transfer through the door opening during a door cycle, a detailed CFD model was created using CATIA V6 which featured modeling of the internal air volume, passenger door interface, external air volume, and the A-class surface of the bus. A photo of the CFD model is shown in Figure 2.

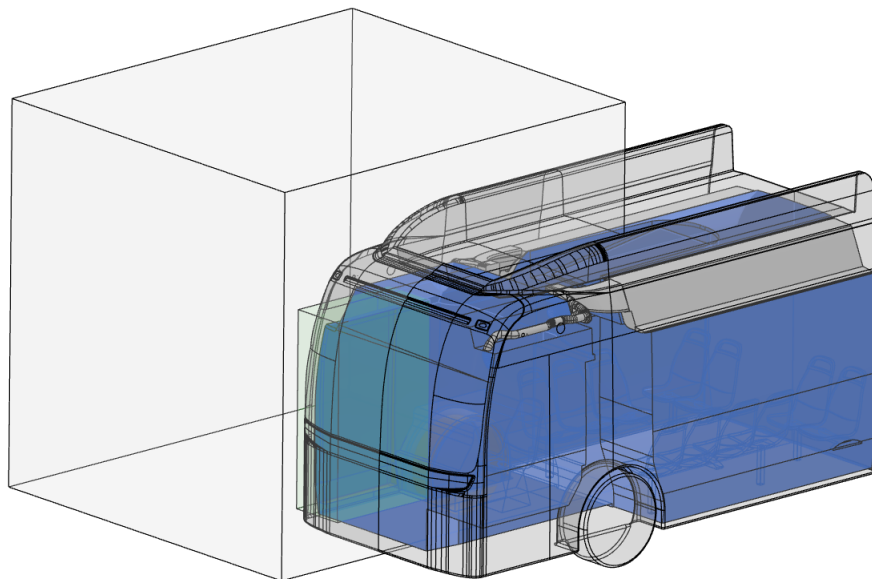


Figure 2: CFD base case model.

The internal geometry of the bus was defeatured to remove passengers, although the seats were left as baffles in the internal volume. The bus was shortened from its 40 ft configuration to 17 ft in order to reduce the cell count of the mesh and increase the computational efficiency of the simulation. The rear door was deleted as the analysis will only focus on the front passenger door, in order to reduce the solving time of the numerical model. The A-class surface was modeled from scratch in order to perform a subtract of the external air volume to ensure the air could not flow in regions where physical bus geometry lies. Additionally, it helps to visualize the separation of the external air and the internal air during post processing of results. The subtract operation output is shown in Figure 3.

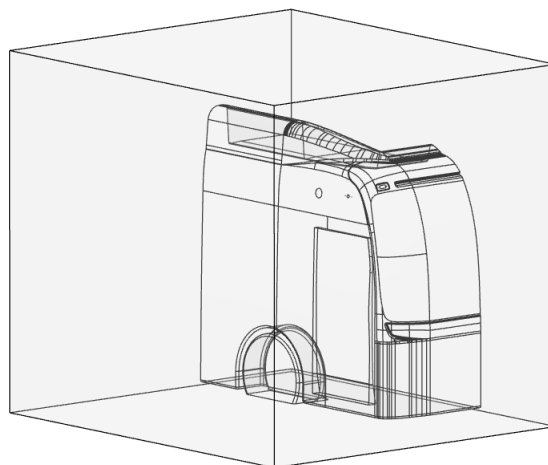


Figure 3: External volume subtract for CFD model, showing the passenger door interface.

The door interface was modeled in the A-class surface and created a proper scale door interface for the internal to external air boundary. This interface for the passenger door is shown in Figure 3. This interface was used to track various relevant heat transfer parameters that will be displayed and discussed in the CFD analysis (Section 4.3). The small green rectangular volume seen in Figure 2 was used to perform manual mesh refinement at the door interface. The mesh will be further analyzed and explained in Section 2.2.

2.2 CFD Methodology

The following section details the methodology used when setting up and performing CFD on the models developed for this project. It explores the software used, meshing, set-up and boundary conditions, and post-processing of the results. The STAR-CCM+ User Guide [3] was consulted regularly during set-up and solution of the various simulations performed.

2.2.1 CFD Software

STAR-CCM+ was used to perform all of the CFD simulation for this project. STAR-CCM+ was selected due to the complex modeling requirements of the problem and the ability to leverage available licences. Additionally, it allowed the team to run multiple simulations on the university computing cluster in order to solve simulations in parallel. Furthermore, STAR-CCM+ has the ability to automatically capture snapshots of the solution at each time step, allowing for automatic data collection and indexing. These screenshots show the entire transient solution over the 7 second period. Following the simulation completion, the screenshots were combined to produce animations of the transient solution and delivered to the client. This is further developed in Section 2.2.4.

2.2.2 Set-up & Boundary Conditions

The pre-processed CAD geometry was saved as a STEP file and imported into STAR-CCM+. Once imported in the into STAR-CCM+, the following process was used to set up the various simulations.

1. Running a Boolean imprint operation to merge common faces.
2. Setting up the solver physics continuum.
3. Splitting the models into required boundary surfaces (split by patch).
4. Assigning the parts to regions

5. Setting initial conditions and specifying boundary conditions and physics values
6. Meshing the model
7. Preparing scenes for post-processing (transient data collection)
8. Running simulation.
9. Modifying time step parameters (inner iterations) based on simulation residuals to improve solving speed.

The implicit unsteady solver was utilized with coupled implicit flow, adaptive time-step and the K-Omega Turbulence model. The implicit unsteady solver was selected as modeling the heat transfer of a door cycle requires the transient solution over a 7 s minimum time step. A solution time of 7 s was selected since this is the minimum door cycle time on New Flyer buses and the heat transfer rate through the door converges and reaches steady-state prior to 7 s of solution. The Implicit unsteady solver controls the update at each physical time step for calculation [3]. Additionally, the time step size and inner iterations are modified used this solver. The inner iterations refer to the amount of iterations performed per time step. For the simulations presented in this report, 10 inner iterations were used as it offered a robust trade-off of computational efficiency and accuracy.

The coupled flow simultaneously solves the conservation equations for mass and momentum using a pseudo-time-marching approach [3]. It was selected as it allows for higher Courant numbers. The Courant numbers are presented with the CFD results in Section 4. The K-Omega Turbulence model was selected due to it's improved performance for boundary layers under adverse pressure gradients [3]. A summary of the solvers used is included in Table II below.

Table II: CFD SOLVER PARAMETERS

Solver :	Implicit Unsteady
Turbulence Model:	K-Omega
Flow:	Coupled Implicit
Under-Relaxation Factor Ramp:	No Ramp

Once the geometry has been imported, and the physics continuum has been specified, the imported parts are split by patch to split and assign names to surfaces of interest (i.e. door boundary, HVAC outlets, pressure outlets etc.). Figure 4 shows the process of splitting by patch. As seen in the figure, the door boundary has already been split and assigned as a separate surface compared to the walls of the internal air volume. This separation is important to allow the assignment of differing boundary conditions to various surfaces. For example, the remaining surfaces shown in Figure 4 are assigned a no-slip wall boundary condition, where as the door boundary is specified as a freestream boundary condition to allow air to flow through the boundary.

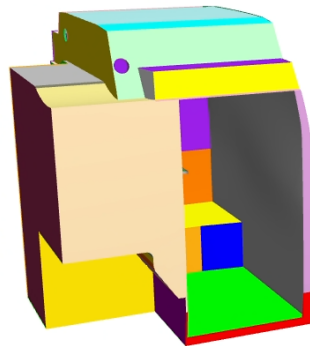


Figure 4: Base case - split by patch operation to split surfaces of interest and create boundaries/interfaces.

Once, the part surfaces have been fully split to separate all important surfaces, the parts are assigned to regions which control the boundaries and physics conditions occurring in these regions. It is important to properly split the part ahead of time, otherwise, the boundaries of interest will not be formed when parts are assigned to the new regions. Figure 5 shows the regions created for the base case simulation below.

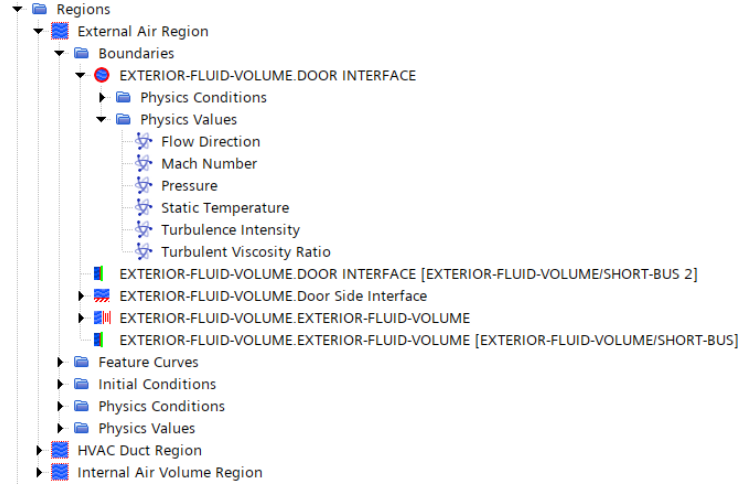


Figure 5: Base case - regions and boundary creation.

The parameters used for each boundary of interest are outlined before the presentation of the simulation results for the base case, optimized base case, and the air curtain in Tables X, XI, and XII respectively.

2.2.3 Mesh

The automated mesh operation was performed in STAR-CCM+. A polyhedral mesh with the automatic surface repair was used to ensure adequate quality cells within the volume mesh. Table III presents the mesh parameters used in the polyhedral mesh below.

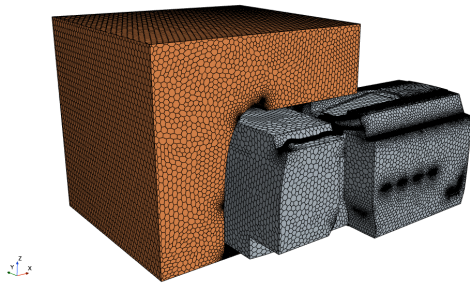
Table III: POLYHEDRAL MESH PARAMETERS USED DURING CFD SIMULATIONS

Mesh Parameter	Value	Unit
Base Size	0.1	m
Minimum Surface Size	5	% of Base Size
Surface Growth Rate	1.3	
Number of Prism Layers	2	
Prism Layer Stretching	1.5	Ratio of each cell layer to the previous layer
Prism Layer Total Thickness	33.33	% of Base Size
Volume Growth Rate	1.2	
Post Mesh Optimization	Optimize Boundary Vertices - Enabled Optimize cell Topology - Enabled	

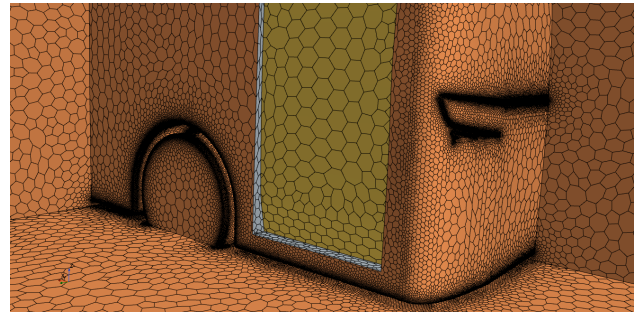
Significant work was required to achieve a conformal mesh at the passenger door interface. Initially, the Boolean subtract of the external air volume was performed in STAR-CCM+, however, this resulted in course surface tessellation at the door boundary surface that matches up to the internal air volume door boundary surface. This resulted in the volume mesh failing or producing a non-conformal mesh. A conformal mesh is defined as a mesh which encompasses multiple parts while the continuity of the mesh remains uninterrupted. [3]. This non-conformal mesh would not allow the solver to execute. Significant time was invested manipulating the subtract settings in STAR-CCM+ before reverting back to performing the subtract in CATIA V6. After re-importing the subtract geometry as perfect CAD data, and experimenting with various mesh parameters (mainly base size and growth ratio) the mesher was able to generate a conformal

volume mesh allowing the simulation to run. Figure 6a shows the surface mesh of the external and internal fluid regions and Figure 6b shows the conformal surface mesh on the passenger bus door interface.

Simcenter STAR-CCM+



(a) Base case - surface mesh.



(b) Base case - conformal door mesh.

Figure 6: Surface mesh for (a) external and internal fluid regions and (b) conformal door mesh.

2.2.4 Post-Processing

Scenes of interest were set up prior to running the final simulations. STAR was programmed to automatically save specific screenshots every ten time-steps (~100 iterations) for the heat transfer coefficient at the door orifice, the air velocity magnitude in the bus cabin and through the door orifice, the heat flux through the door, and the temperature gradient through the bus cabin and external air (ambient). This automated photo capture feature allows the transient response throughout the 7 s to be characterized and animated to visualize the solution. Over 70 GB of post-processed files have been gathered and will be delivered to our client.

3 Concept Development, Analysis and Selection

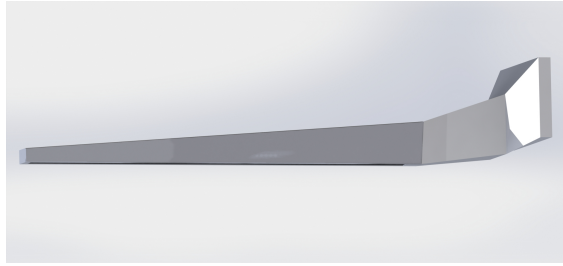
In this section, different aspects of the design needed to be considered when developing the air curtain. The major aspects considered are the plenum design, fan requirements to provide appropriate CFM, and nozzle exit velocity. These aspects are analyzed in the following sections.

3.1 Plenum Designs

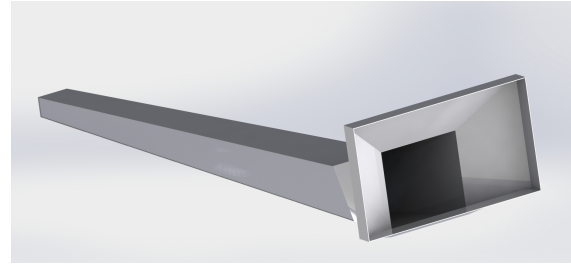
The Plenum is a critical part of the air curtain as it directs the flow from the fan to the nozzle exit. Carefully designing the plenum will allow laminar flow to develop before it exits the air curtain to effectively cover the door opening. By brainstorming different methods of fan placements in the unused space around the door mechanism, three different designs were selected and compared using a matrix. The four designs considered includes a side fan (design #1), dual fan (design #2), top mount fan (design #3), and a off-the-shelf air curtain (design #4). The following sections outline the four designs.

3.1.1 Design #1

This concept makes use of a single fan placed at the side making use of the unoccupied space at the side of the door. Placement of the fan away from the exit allows the turbulent flow from the fan to transition to laminar flow as it travels through the plenum. The flow is accelerated through the plenum due to the narrowing cross-section of the structure which counters the pressure drop experienced as the flow moves towards the exit. This effectively covers the entire door opening. Thus, it is important to have a sufficient velocity of the air at the exit of the air curtain. Figure 7 shows the plenum design.



(a) Side view.

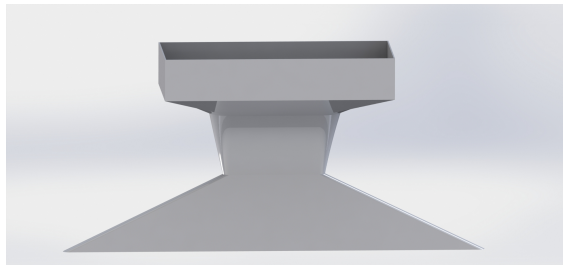


(b) Front view.

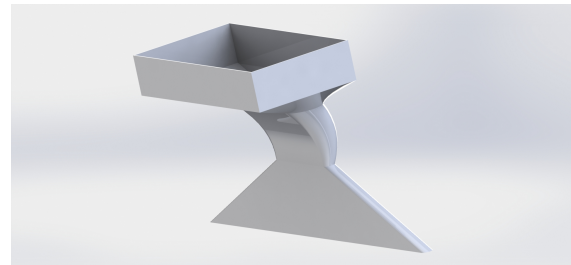
Figure 7: Design #1 renders.

3.1.2 Design #2

This concept makes use of two axial fans to meet the CFM requirement of the air curtain. The axial fans are housed inside the plenum beside each other and are placed above the door mechanism to make use of the available space. It also ensures that the fans are placed away from the exit of the nozzle to allow the airflow to develop a laminar flow. The top of the plenum is a rectangle-shaped design to accommodate the fans, and the midsection of the plenum is a slot-shaped curved design which is chosen because of the tight space between the door mechanism and the side panel and also to provide clearance to the door mechanism. As the airflows down the plenum, its velocity increases due to the reduction in cross-sectional area. The bottom section of the plenum has a wider cross-sectional area which is designed to meet the airflow speed requirements for the air curtain to cover the total width of the door opening.



(a) Front view.



(b) Side view.

Figure 8: Design #2 renders.

3.1.3 Design #3

This plenum was surface modeled in CATIA V6 and was packaged around existing bus systems including the door mechanism and surrounding components. The geometry was designed to be produced using rotational molding (rotomolding), allowing complete freedom of the geometry to be smooth and organic to promote optimum outlet flow. Due to the lower production volumes of buses, rotomolding was selected instead of injection molding. Figure 9 shows a top view of the plenum in the passenger door assembly below.

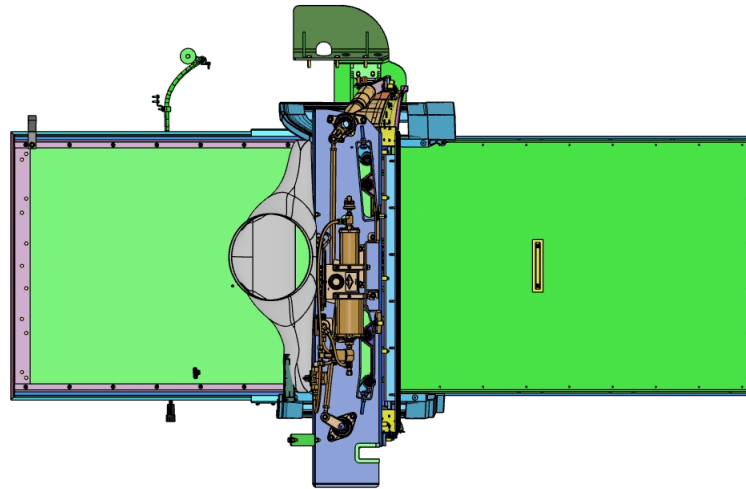
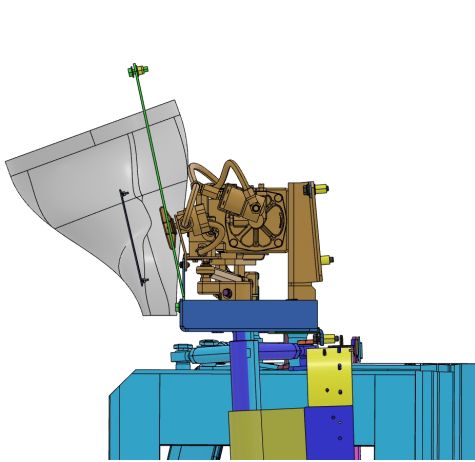
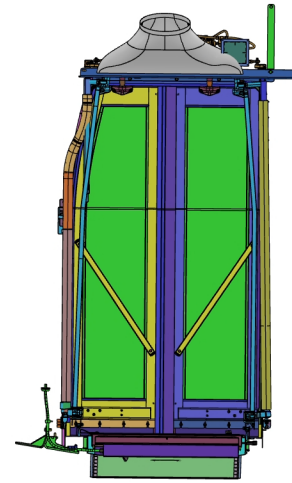


Figure 9: Design #3 - plenum top view in passenger door assembly.

A side view of the plenum is displayed in Figure 10a, and a front view of the plenum and door assembly is presented in Figure 10b.



(a) Side view of plenum showing packaging with the existing door mechanism.



(b) Front view of plenum and door assembly.

Figure 10: Design #3 - packaging of plenum with existing passenger door assembly.

Final renders of the design are presented below in Figure 11.

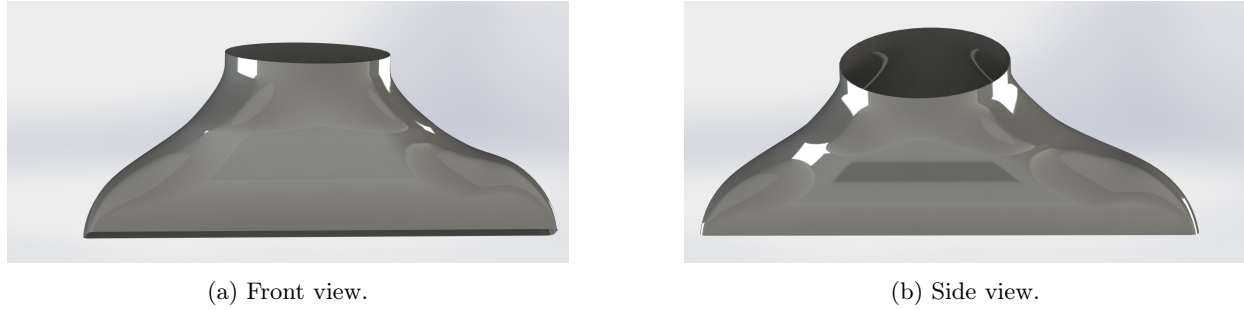


Figure 11: Design #3 renders.

Section 5.1 shows a detailed CFD analysis of this plenum design in the context of the bus.

3.1.4 Design #4

Another option is to purchase an off-the-shelf air curtain. NFI can reduce engineering costs by purchasing an off-the-shelf option from a supplier that would satisfy the requirements. NFI outsources numerous parts of the bus and therefore could be implemented. Figure 12 shows an air curtain model OPT 2000 A from airtecnic that meets the requirements of the project.



Figure 12: Off-the-shelf air curtain.
[4]

3.2 Concept Screening

Table IV shows a weighted decision matrix (WDM) on the four plenum designs. This tool utilizes selection criteria based on the the customer needs and target specifications and are given a weight based on their importance. The team used airflow, packaging, noise, weight and cost as the five most important selection criteria. The higher the weight the greater the importance. Each of the four designs are given a score from one to five where one meant it performed poorly and five meant it performed best for the given metric. Using these weight and scores, a total weighted score was determined and final design selection was based on this scoring.

Table IV: PLENUM SELECTION METRIC

		Concepts							
		Design #1		Design #2		Design #3		Off-the-shelf	
Selection Criteria	Weight	Rating	Weighted Score	Rating	Weighted Score	Rating	Weighted Score	Rating	Weighted Score
Airflow	40%	3	1.2	3	1.2	4	1.6	5	2
Packaging	25%	4	1	3	0.75	3	0.75	2	0.5
Noise	15%	2	0.3	3	0.45	4	0.6	4	0.6
Weight	10%	4	0.4	2	0.2	4	0.4	2	0.2
Cost	10%	4	0.4	2	0.2	3	0.3	2	0.2
Total Score		3.3		2.8		3.65		3.5	
Rank		3		4		1		2	
Select		NO		NO		YES		NO	

Energy efficiency is the most important client need and it is achieved by reducing heat loss through the door. Having low velocity or highly turbulent airflow would reduce the potential benefit of the air curtain is the quality of the barrier would be reduced. Therefore, it is given the highest weight in the WDM.

Packaging refers to how easily the air curtain can be installed into the available space above the door and if it would require any design changes to the existing bus parts. Through a discussion during a meeting it was determined that the client wants to avoid making any changes to the existing parts as that could lead to scope creep. Hence, packaging criteria is weighed the second highest in the WDM.

The noise created during the operation of the air curtains is a minor factor in the selection criteria of the plenum design and that reflects the reduced weight given to it. This is due to the fact that the air curtain will operate in during passenger loading when it is often already loud.

Weight and cost were both weighted the lowest. The reason is that the air curtains weight depends mostly the weight of the fans and less on plenum. Furthermore, if the fans and plenum are both securely mounted then their weight would not be a major issue. The client's focus is centred around finding a solution to the problem and clearly stated that cost is not a major concern within the scope of this project. Thus, cost was given a low weight.

Design four was given the highest score of 5 for the airflow as it as an off the shelf air curtain that is operated by centrifugal fans that produce a smooth laminar flow, which a major factor in having the air curtain operate at its full potential. Design 3 is given a score of 4 because it is operated by an axial fan that produces a turbulent flow but the plenum design allows the airflow to develop into a laminar flow. Designs one and two are both given lower scores of 3 because they are operated by an axial fan, but the plenum design does not allow the airflow to completely develop into a laminar flow. This implies means a mixture of turbulent and laminar flow exiting the air curtain which reduces the benefit of having an air curtain.

Design one scored 5 in the packaging category. This is because it is designed in a manner such that the fans are mounted on the side of the door mechanism and the plenum sits below the door mechanism where there is enough space for the plenum to fit in without requiring any changes to the existing parts. Designs two and three both are given a score of 3 because both the designs require changes to existing parts of the bus interior. Design four is an off the shelf air curtain which is not designed specifically for implementation in the bus door system, therefore, it would require major changes to the existing parts of the bus and is given a score of 2.

Design 4 scores the highest (5) in noise criteria and the major reason is that it is operated by centrifugal fans which are quieter than the axial flux fans that the other three designs are using. Design one is given the lowest score of two, because the plenum is designed such that it causes sudden changes in the surface area of the plenum which creates high pressure gradients causing the airflow to hit hard on the inner walls of the plenum producing noise. Design two and three score 3 and 4 respectively in the noise criteria as both the plenum designs allows for a smooth transition from higher to lower cross sectional area of the plenum which produces less noise.

In weight and cost criteria, both design one and design three performs well and are given a score of 4 while design two and design four performs poorly and are given a score of 2. Designs one and three plenums use less material and a single fan and therefore, weigh and cost less compared to design two and four which uses more plenum material and double fans.

The total weighted scores of the designs ranked them 3, 4, 1, 2 from highest to lowest. This makes sense because design three uses a single axial fan with a smooth plenum design allowing the airflow to smoothly transition from turbulent to laminar flow without large pressure changes. It is comparatively lighter than other designs because it is operated by single fan and since it is designed especially to fit into the available space with minimal design changes it packages well. Additionally, by using less plenum material for this design it makes it cost effective.

Note that design three was still used for the final design but the method of powering it was changed to a dual centrifugal setup.

3.3 Fan Selection

Efficiency and airflow of the fan(s) used in the air curtain are the two most important factors when selecting a fan for this application. This section will cover the different types of fans considered for use in the air curtain.

3.3.1 Axial Fan

Axial fans are designed to direct the flow in the axial direction as it draws air. Axial fans are efficient at producing a high volume of air movement while using up less power making them affordable to operate [5]. Axial fans are compact making them useful in applications like this one where space is limited [6]. For these reasons the axial fan is a viable option for the final design.

3.3.2 Tangential Fan

Tangential fans, also known as cross-flow fans, are commonly used in existing air curtain designs as they provide a uniform airflow across the length of the air curtain. However, the low airflow velocity produced by these fans makes them less favorable in air curtains where higher air velocity requirements exist. An advantage of these fans is that they are quieter than alternative fan choices. Ultimately, this fan type was not considered as it did not provide the airflow velocity that is required to seal the bus door in its harsh working environment.



Figure 13: Tangential fan.
[7]

3.3.3 Centrifugal Fan

Centrifugal fans are designed to produce airflow perpendicular to the blower axis [8]. Centrifugal fans are widely used in the heating and cooling process and electronic cooling as they are capable of producing high pressure flow with low flow rate [8]. Due to low flow rate produced by the fan, a combination of 2 or 3 fans are used in the air curtain to produce the required airflow and as a result end up using more space and making them less viable option when space is limited. Figure 14 shows the centrifugal fan used in the air curtain.



Figure 14: Centrifugal fan.

Three different fans were considered for the design application, axial, tangential, and centrifugal fan. A final decision was made to go with the axial fans as they have an advantage of being used in smaller spaces while providing high velocity flow rather than high pressure.

Note that this decision was altered in the final design of the air curtain as analysis proved that a dual centrifugal fan setup would work best.

3.4 Exit Velocity

The velocity of the air exiting the nozzle of the plenum was a key design parameter. Not having sufficient airflow will result in premature flow deviation, resulting in incomplete coverage of the doorway. Furthermore, having faster flowing air will reduce the effects of any transverse pressure the air curtain must overcome from pressure build-ups internally or wind conditions externally. Previous studies done on various applications of the air curtain, such as residential buildings and commercial buildings, acted as the reference point to determine the exit velocity for bus application. Work done by Chang shu et al with the air curtain for a 95 inch door opening demonstrated that air speed in the range of 8-18 m/s exit velocity is required from the air curtain [9]. This is also supported by The Engineering ToolBox indicating the air discharge from the above are found to be in the range of 5-15 m/s [10]. Based on the research, the team decided to select 15m/s flow velocity for the air curtain. This was used as a starting point and simulation was carried out using CFD.

4 Analysis

This section will cover analytical and numerical methods of analyzing heat transfer. Determining the behavior of airflow and heat transfer over time is a complex problem, and CFD analysis is the primary tool used for this investigation. Analytical calculations are carried out to approximate the heat transfer through the door to compare the results with those obtained from the numerical approach to help validate the numerical model used.

The team received information from the client that contains daily Average HVAC Loads for its fleet which is summarized in Table V. Note that the team received detailed information for the entire fleet but the table below only shows the key data extrapolated by the team. This extrapolation was done to summarize the fleet as a whole using the daily average fleet HVAC loads and the most extreme conditions with Winnipeg, MB being the coldest and Pheonix, AZ being the warmest.

Table V: SUMMARIZED DAILY HVAC LOADS FOR NFI FLEET

REGION	BUS SUM	HVAC DAYTIME HEATING LOAD (kW)								HVAC COOLING LOAD (kW)												AVG HEAT (kW)	AVG COOL (kW)
		JAN	FEB	MAR	APR	OCT	NOV	DEC	JAN	FEB	MAR	APR	MAY	JUNE	JUL	AUG	SEP	OCT	NOV	DEC			
FLEET AVERAGE	16866	4.98	4.28	2.41	0.46	0.22	1.77	4.00	0.36	0.45	0.66	1.34	2.87	4.57	5.54	5.30	3.94	1.88	0.69	0.36	1.51	2.33	
WINNIPEG, MB	253	14.92	13.14	8.70	2.78	2.48	8.40	13.44	0.00	0.00	0.00	0.00	1.40	3.52	4.57	4.15	1.82	0.00	0.00	0.00	5.32	1.29	
PHEONIX, AZ	280	0.00	0.00	0.00	0.00	0.00	0.00	0.00	1.61	2.46	3.94	5.84	7.96	10.07	10.71	10.29	9.02	6.48	3.31	1.40	0.00	6.09	

4.1 Energy Consumption Calculations

From the daily HVAC loads presented in Table V, the daily energy consumed by the battery for HVAC purposes can be calculated for the highest heating load (Winnipeg, MB), highest cooling load (Pheonix, AZ), and the fleet average respectively. The Coefficient of Performance (COP) for heating is 1 while the COP of cooling is between 2.3 and 3.5. For the purpose of these calculations the lower COP value of 2.3 for cooling will be used. An assumed daily average operating time of 12 hours, which was agreed upon by the client, is used. Using this assumption, the energy consumed due to HVAC loads can be calculated using Equation 1. Equation 1 explains the relationship between power and energy, where E is energy (kWh), P is average instantaneous power (kW), and T is time (h) respectively.

$$E = P * T \quad (1)$$

The calculations for highest average heating load are shown below in Equation 2:

$$E_{heat} = P_{heat} * T \quad (2)$$

$$E_{heat} = 5.32 * 12$$

$$E_{heat} = 63.84 kWh$$

Therefore, in Winnipeg, the New Flyer buses use an average 63.84 kWh of battery capacity due to heating the bus passenger compartment. In comparison, the energy consumed is far greater in January when the buses draw an average daily power of 14.92 kW. The energy consumed is computed below in Equation 3, where E_{heat*} represents the energy consumed in Winnipeg during the extreme month of January.

$$E_{heat*} = P_{heat*} * T \quad (3)$$

$$E_{heat*} = 14.92 * 12$$

$$E_{heat*} = 179.04 kWh$$

This results in a daily energy consumption of 179.04 kWh in January. This is a significant portion of the battery that is being used exclusively to maintain cabin set point temperature. New Flyer currently offers two different battery capacities 525 kWh and 660 kWh. This represents the total energy available for the packs, but in reality there is approximately 15% capacity that is dedicated to a buffer to prevent wear and tear, ensure core bus function and to increase the longevity of the pack. The adjusted battery capacity is presented in Table VI.

Table VI: ADJUSTED BATTERY CAPACITY AND EFFICIENCY OF NEW FLYER ELECTRIC BUSES

Battery Capacity (kWh)	Adjusted Battery Capacity (kWh)	Bus Range (km)	Bus Efficiency (kWh/km)
525	446.25	402	1.11
660	561	505*	1.11*

Note: The * indicates that the data is unofficial and calculated from extrapolation.

The battery consumption ($B_{consumption*}$) due to heating in January is calculated in Equation 4 below.

$$B_{consumption*} = \frac{E_{heat*}}{E_{capacity}} \quad (4)$$

$$B_{consumption*} = \frac{179.04(kWh)}{446.25(kWh)} * 100$$

$$B_{consumption*} = 40.1\%$$

Therefore, the daily HVAC in January will consume approximately 40.1 % of the usable battery capacity of the bus. Given that the quoted range for the battery electric bus with 446.25 kWh of usable capacity is 402 kilometers. The range reduction due to heating consumption can be calculated using Equation 5. Where R_{loss} represents the range loss per full charge due to heating the passenger cabin.

$$R_{loss*} = \frac{R}{E_{capacity}} * E_{heat} \quad (5)$$

$$R_{loss*} = \frac{402(km)}{446.25(kWh)} * 179.04(kWh)$$

$$R_{loss*} = 160.4km$$

Therefore, in the month of January, buses in Winnipeg consume on average 179.04 kWh of energy daily which equates to 40.1% of the overall battery capacity and a bus range reduction of 160.4 km per charge. The results of the preceding calculations are tabulated in Table VII below.

Table VII: HVAC ENERGY CONSUMPTION IN NEW FLYER BUSES

	Energy Consumption (kWh)		Battery Capacity Used (%)	Range Loss (km)
	HVAC Heating Loss	HVAC Cooling Loss		
Winnipeg Avg	63.84	6.73	15.8	63.6
Winnipeg January (*)	179.04	0	40.1	160.4
Phoenix Avg	0	31.77	7.1	28.6
Phoenix July (*)	0	55.88	12.5	50.3
Fleet Avg	18.12	12.16	6.8	27.3

The data tabulated in Table VII demonstrates the large effect climate has on the overall energy consumption of the HVAC systems. With the majority of losses occurring in extreme temperatures as expected. The maximum energy consumption based on the available data occurs in January in Winnipeg. This is a HVAC loss of 179.04 kWh, resulting in 40.1% of the onboard battery being used for heating exclusively and a 160.4 km reduction in the overall bus range. The second worst scenario is in the month of July in Phoenix, AZ. Phoenix buses use on average 55.88 kWh worth of AC daily, resulting in 12.52% worth of usable battery capacity used solely for AC and 50.3 km of range lost due to the reduced battery capacity per charge. Higher energy consumption in January in Winnipeg also results in overall higher HVAC loss in an year in Winnipeg compared to HVAC loss occurring in extreme condition in month of July in Phoenix.

4.2 Heat Transfer Through the Door

It is important to to reduce the amount of energy consumed by the HVAC to maintain the cabin set point temperature. There are several factors contributing to the heat loss and in order to develop an optimal solution to the heat loss it is crucial to accurately determine the various heating loads. Note, in this project we are only concerned with the heat transfer through the door of the bus when it is open and closed.

4.2.1 Closed Door

Heat transfer through the closed door is a combination of conduction through the thickness of the door, which is made of glass with an aluminium frame, and convection between the inner air and the inner surface of the bus and the exterior air and the exterior surface of the bus.

This section will calculate the heat transfer through the door in the most extreme conditions with Winnipeg, MB being the coldest in January with an average outside temperature of -18°C [11]. Phoenix, AZ is the hottest in July with an average outside temperature of 41°C [12]. The cabin set point temperature is assumed to be 15°C in this calculation. Using Equation (6) heat transfer through the closed door can be calculated as [13].

$$Q = A * U * (T_i - T_0) \quad (6)$$

Where A is the cross-sectional area (m^2), U is the heat transfer coefficient ($\frac{W}{m^2K}$), T_i is the cabin temperature of the bus (°C) and T_0 is the ambient temperature (°C). Heat transfer coefficient is calculated using Equation (7) [13].

$$\frac{1}{U} = \frac{1}{h_i} + \frac{L}{k} + \frac{1}{h_o} \quad (7)$$

Where h_i is the convective heat transfer between the cabin air and the inner surface of the bus ($\frac{W}{m^2K}$), h_o is the convective heat transfer between the ambient air and the external surface of the bus ($\frac{W}{m^2K}$), k is the conductive coefficient of the solid material and L is the thickness of the material. Convective heat transfer coefficient h_i and h_o are calculated using Equation (9) [13].

$$h = 9 + 3.5 * v^{0.66} \quad (8)$$

Where v is the speed of air relative to the surface. Convection between the outside surface and the outside air depends on the velocity of the bus while the convection between the interior surface and interior air depends on the velocity of the interior air.

To calculate the heat transfer through the closed door, cabin air is assumed to be still and therefore, its velocity is taken as 0 (m/s), which results in $h_i = 9$ ($\frac{W}{m^2K}$). In Winnipeg, bus travels at 16.67 m/s and in Phoenix it travels at 11.11 m/s resulting in $h_{0,Winnipeg} = 31.42$ ($\frac{W}{m^2K}$) and $h_{0,Phoenix} = 26.15$ ($\frac{W}{m^2K}$) [14]. Glass section of the bus door is 0.02 (m) thick and its cross-sectional area is 1.109 (m^2) while conductive heat transfer k of glass is 1.05 ($\frac{W}{mK}$). The aluminum frame is 0.0254 (m) thick and its cross sectional area is 0.517 (m^2) while the conductive heat transfer k of aluminum is 239 ($\frac{W}{mK}$).

The door opens for 7 seconds at each stop and according to the Winnipeg transit website, the blue bus stops 47 times per trip [15]. The bus makes one complete trip in 40 minutes and assuming that the bus operates for 12 hours in a day, resulting in each bus completing 18 full trips per day. The total time the bus door remains closed is 43200 seconds and open for 5922 seconds. Using Equations (6) and (7) we can calculate the total heat transfer through the door in two extreme conditions. Table VIII shows the results from the calculations:

Table VIII: CLOSED DOOR HEAT TRANSFER SUMMARY

	Heat Transfer Through Closed Door (kWh)
Winnipeg (January)	7.70
Phoenix (July)	5.81

4.2.2 Open Door

Heat transfer process through the open door is difficult to estimate because of the complexity of the flow of the outside air. The airflow pattern of the outside air depends on weather conditions, wind speed and its direction which continuously changes. To understand how the heat transfer process works, we will assume the air inside the bus to be ideal and compressible. When the door opens, the outside air moves inside the bus which pushes onto the inside air and compresses it to accommodate the volume of incoming outside air. This pressurizes the inside air, however, when the air enters the bus it mixes with the inside air and moves out of the bus because of the change in airflow direction, the temperature difference between the inside and outside air and the pressure created inside the bus and thus causes heat transfer.

To help simplify the calculation the team used the study done on the airflow into the room through a small crack inside the wall [16]. Parameters used in this study were the opening size of the crack which is equal to $0.05m^2$, opening depth of 0.2 m room volume of $1000m^3$ outside and inside temperature of 0 ($^{\circ}C$) and 22 ($^{\circ}C$), respectively, and wind velocity of 10m/s. Figures 15 - 16 shows how each of these factors effects the airflow rate when the other parameters are unchanged.



(a) Flow rate vs enclosure volume.



(b) Flow rate vs opening depth.

Figure 15: Flow rate relation to enclosure volume and opening depth.
[16]

(a) Flow rate vs wind velocity.

(a) σ_q and fitting vs. Opening size
(b) Opening size vs flow rate.Figure 16: Flow rate relation with wind velocity and opening size.
[16]

To have a better approximation of the airflow rate through the open door we will determine the four different airflow rate values from each of these graphs, add them to determine an approximate value of the airflow rate through the bus door. Heat loss through an open door can be calculated using the Equation (9):

$$Q = V * \rho * c * (T_i - T_0) \quad (9)$$

This section will calculate the heat transfer through the door in the most extreme conditions with Winnipeg, MB being the coldest in January with average outside wind speed of 4.83 (m/s) [11]. Phoenix, AZ being the hottest in July with average outside wind speed of 3.14 (m/s) [12]. The bus door Opening size is 1.63 (m^2), opening depth is 0.0254 (m) and internal bus volume is 64.28 (m^3). airflow rates are calculated from each of the above graph and added to determine the approximate value for the airflow rates which are 0.457 (m^3/s) and 0.447 (m^3/s) for Winnipeg and Phoenix, respectively.

Table IX: OPEN DOOR HEAT TRANSFER SUMMARY

	Heat Transfer Through Closed Door (kWh)
Winnipeg (January)	56.81
Phoenix (July)	43.79

The heat transfer results from the analytical analysis is far below the average HVAC load data of the buses we got from the client, this is due to several assumption that were made to simplify the calculations, therefore, we will refer to the clients provided data for this report

4.3 Base Case CFD Analysis

The base case numerical analysis captures the current performance of the new flyer door through a 7 s (minimum door cycle time) transient opening. The 1500 CFM system used on 40 ft buses was divided amongst the HVAC outlet ducts in the front half of the bus. For the base case, the model was halved in order to reduce the size of the mesh and reduce the solving time and improve the computational efficiency of the simulation. After speaking with the client, a mass flow rate of 200 CFM was assigned to the HVAC air duct outlets as a reasonable assumption given the total 1500 CFM system utilized on a 40 ft bus.

4.3.1 CFD Parameters

The CFD simulation parameters for the base case are shown in Table X below.

Table X: BASE CASE - CFD SIMULATION INPUT PARAMETERS AND BOUNDARY CONDITIONS

Region	Boundaries/Interfaces	Boundary Condition	Value	Unit	Region Initial Temperature	Unit
External Air Region	Door Interface	Freestream	N/A	N/A	253	K
	Door Side Interface	Wall (No Slip)	N/A	N/A		
HVAC Duct Region	Driver Front HVAC Inlet	Mass Flow Inlet	0.1159	kg/s	318	K
	Driver Rear HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
	Passenger Front HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
	Passenger Rear HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
Internal Air Region	Door Interface	Freestream	N/A	N/A	288	K
	Internal Fluid Volume Walls	Wall (No Slip)	N/A	N/A		

The parts were assigned to three separate fluid regions including the external air region, the HVAC duct region, and the internal air region. Due to the implicit unsteady solver being used, the initial region temperature was specified as 253 K (~ -20 C) for the external air, 318 K (~ 45 C) for the HVAC temperature, and 288 K (~ 15 C) as the internal air temperature respectively. The external air temperature was selected to simulate a cold day in the month of January in Winnipeg. This would show the most benefit from reducing heat loss as this month exhibits the largest losses due to heating cost showed in Section 4.1. For boundary conditions, the door interface was specified as freestream, meaning there is no resistance for the flow to pass through the boundary. The four HVAC inlets were specified with a mass flow rate of 0.1159 kg/s based on the 200 CFM assumption. The walls of the internal fluid region were given a wall (no slip) boundary condition.

4.3.2 Velocity Streamlines

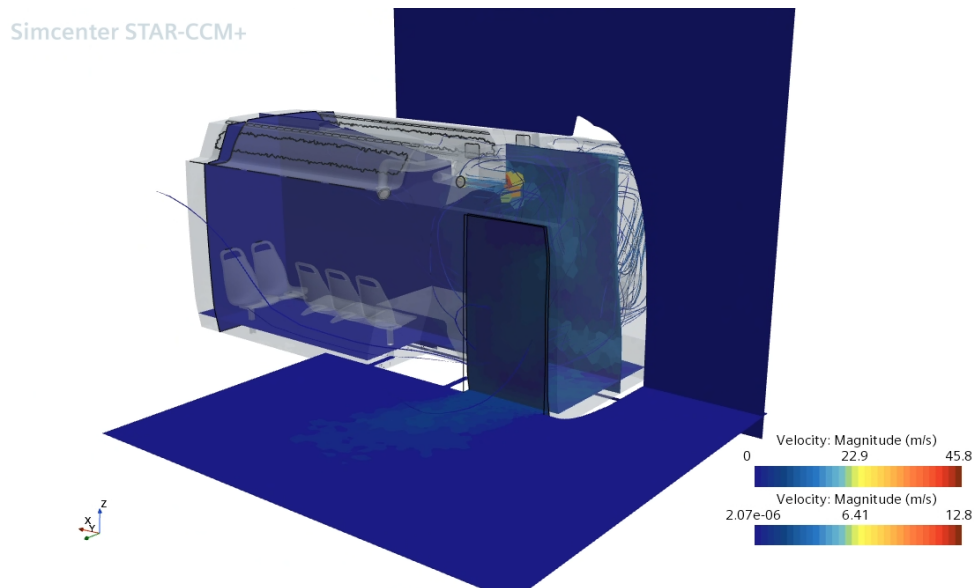


Figure 17: Base case - velocity streamlines through door boundary.

Figure 17 shows the velocity streamlines through the door boundary for the current New Flyer door system after 7 s of transient solution. The upper velocity scale belongs to the streamlines and the lower scale the velocity section planes respectively. The velocity streamlines show the flow swirling in the front of the bus near the bus entrance and eventually leaving through the passenger door boundary. The velocity gradient is as expected and the flow velocities leaving the HVAC duct outlets seem reasonable for a passenger bus. The streamlines show the general flow gradients near the front passenger door and help visualize where in the door boundary the highest losses due to heat transfer occur.

4.3.3 Temperature Gradient

Figure 18a shows the temperature gradient 1 s after the door opens and Figure 18b shows the temperature gradient after 7 s of transient solution respectively.

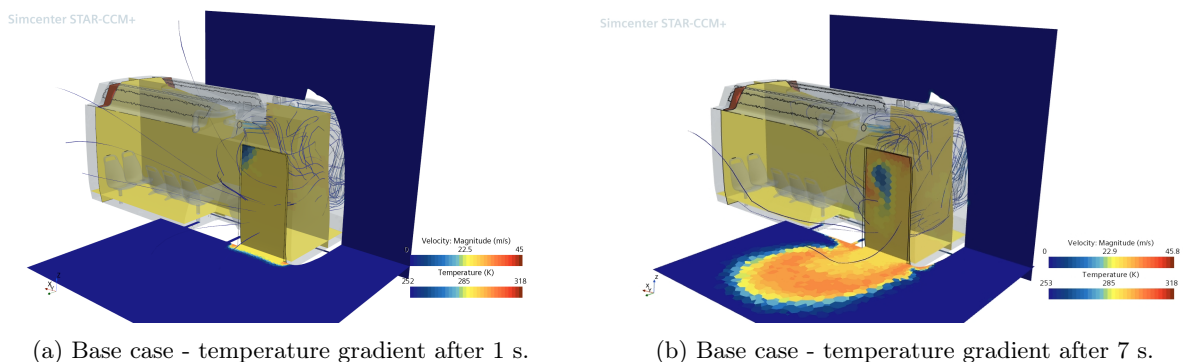


Figure 18: Base case - temperature gradients through door boundary and propagation into external air medium for (a) 1 s and (b) 7 s of transient solution.

It is evident that immediately after the door opens, the internal air begins to exit and after 7 s, a significant volume of air has exited the bus. Through examination of the cells around the door opening, the temperature

abruptly changes in Figure 18a, where as the temperature gradient in the air volume directly outside the door in Figure 18b has a smoother transition and more linear decrease in temperature as the cells get further from the bus door. This behavior is expected and shows the importance of limiting the amount of time the bus door is open when the bus is stopped. The longer the door is open, the more energy will escape through the door. The following figure presents the heat transfer coefficient through the passenger door boundary.

4.3.4 Heat Transfer Coefficient

The heat transfer coefficient for the door boundary is presented in Figure 19.

Simcenter STAR-CCM+

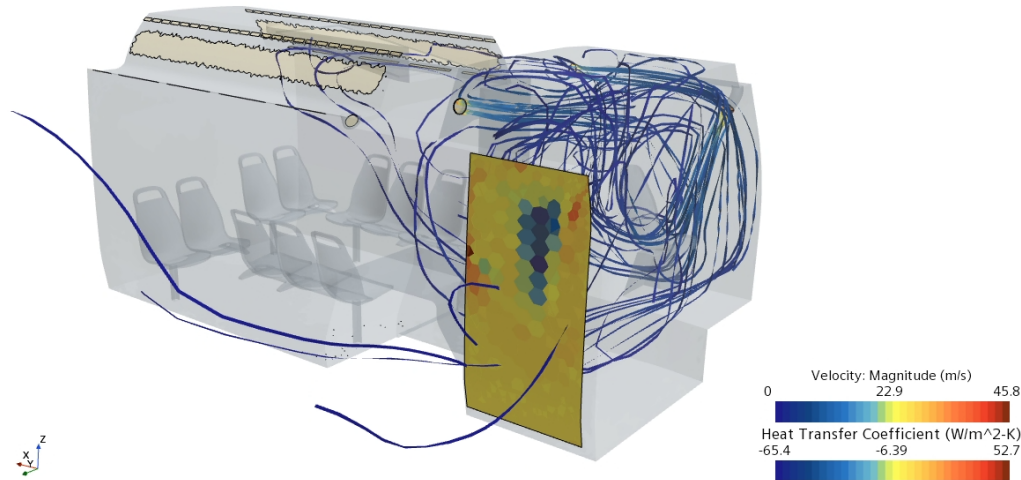


Figure 19: Base case - heat transfer coefficient through door boundary.

STAR-CCM+ uses the boundary heat flux field function to calculate the heat transfer coefficient [3]. The heat transfer coefficient describes the ability for heat to be conducted through the boundary. At 7 s, the heat transfer coefficient surface average at the door boundary is $1.781 \text{ W/m}^2\text{K}$

4.3.5 Boundary Heat Flux

Figure 20 shows the boundary heat flux through the door boundary. Boundary heat flux is the magnitude of the heat flux vector normal to the boundary [3]. Positive values indicate heat is flowing out of the domain [3]. The average surface value of boundary heat flux at 7 s of transient solution is $6.36 \times 10^4 \text{ W/m}^2$. Therefore, in this base case, $\sim 63,600 \text{ W}$ per square meter of area is exiting through the door boundary.

Simcenter STAR-CCM+

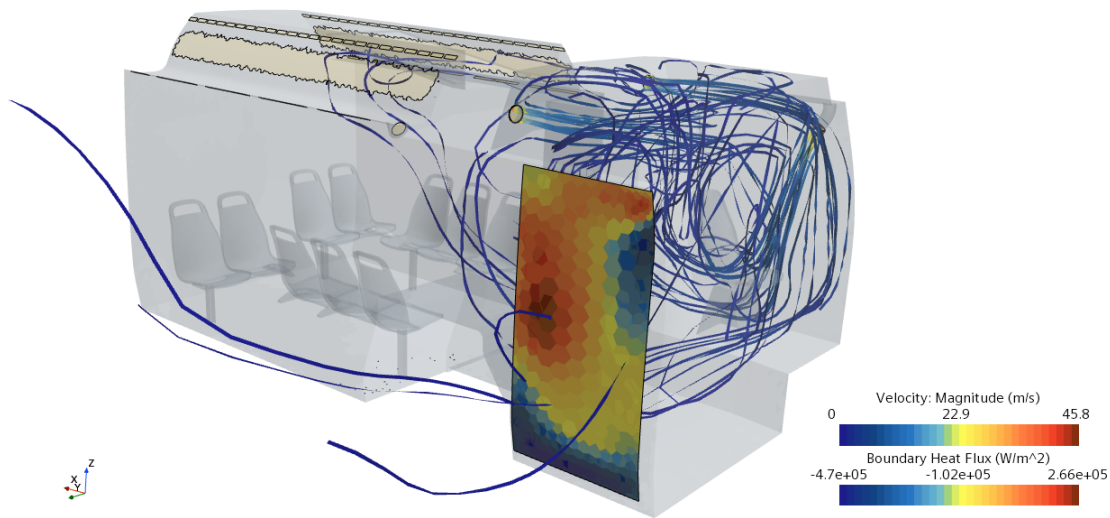


Figure 20: Base case - boundary heat flux through door boundary.

4.3.6 Mass Flow Rate Through Door Boundary

The heat transfer through the door boundary is directly proportional to the amount of conditioned cabin air that crosses the door boundary. Figure 21 presents the transient mass flow rate response over a 7 s door cycle.

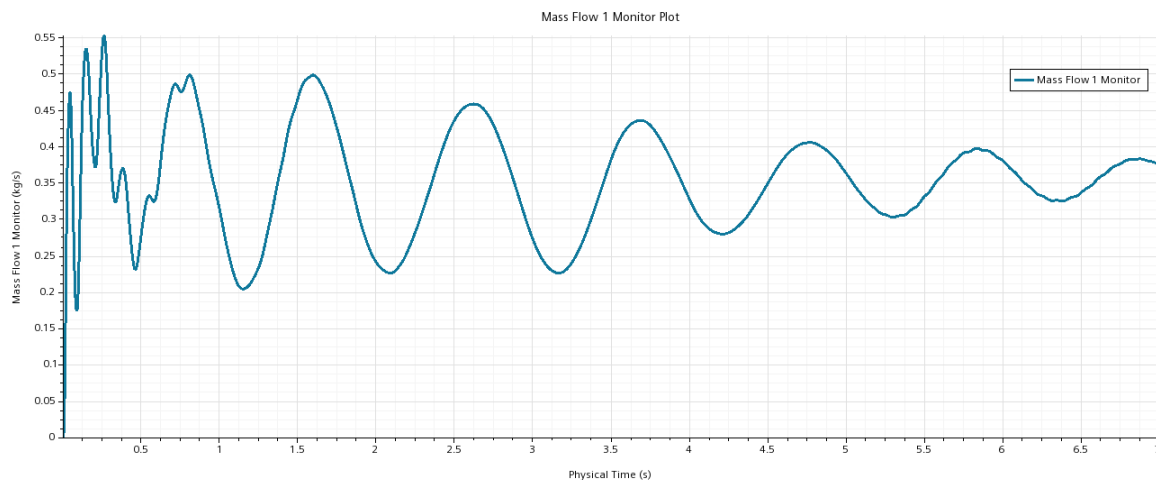


Figure 21: Base case - transient mass flow rate through door boundary.

The function is identical to the transient heat transfer rate response of the system shown in the following section.

4.3.7 Heat Transfer Through Door Boundary

Figure 22 presents the transient heat transfer response over the 7 s door cycle. The value is oscillating, however appears to converge to ~ 105 kW of instantaneous heat transfer rate through the door boundary for loading periods greater than 7 s.

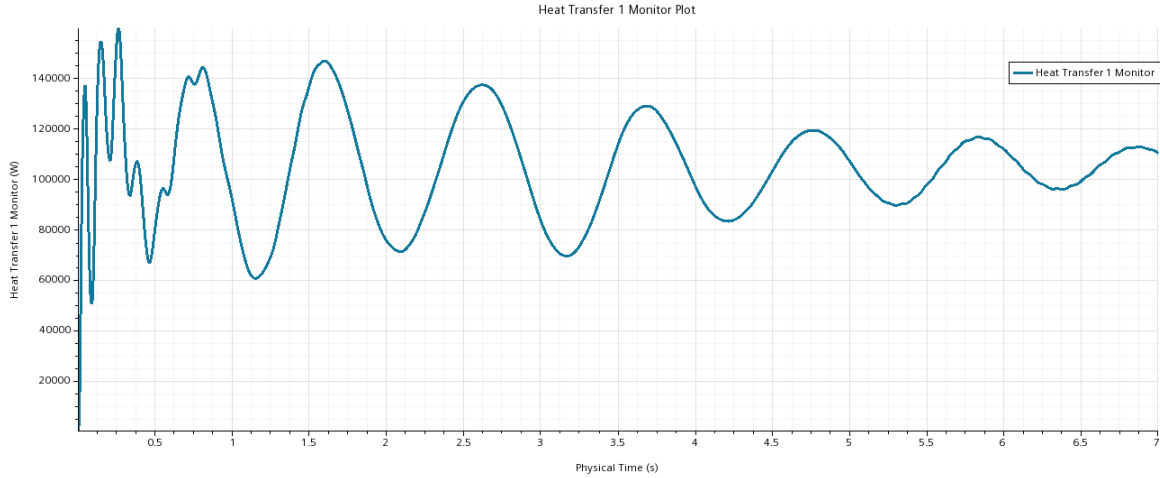


Figure 22: Base case - transient heat transfer response through door boundary.

The value is oscillating, however appears to converge to ~ 105 kW of instantaneous heat transfer rate through the door boundary for loading periods greater than 7 s. For longer multi-passenger loading situations, the heat transfer rate would likely remain steady around 105 kW.

4.3.8 Wall Y^+ Histogram

Figure 23 presents the Wall Y^+ values for each boundary for the base case simulation.

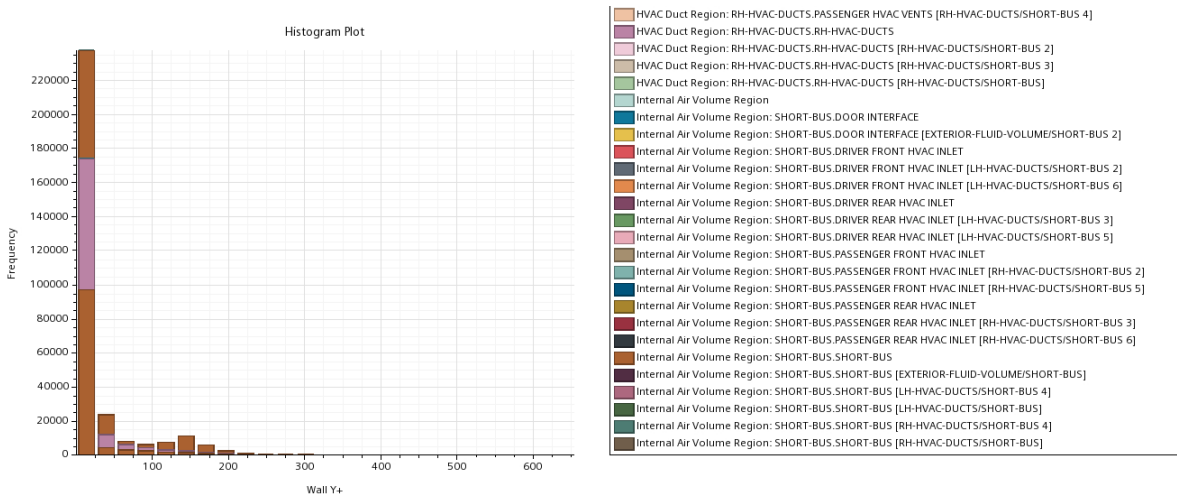


Figure 23: Base case - wall Y^+ histogram.

Wall Y^+ is the non-dimensional wall adjacent grid height [17]. Since we are dealing with a fairly low Reynolds number, the Wall Y^+ tabulated in Figure 23 is within a reasonable range.

4.3.9 Convective Courant Number Histogram

Figure 24 shows the convective Courant number for all of the regions and boundaries in the base case simulation.

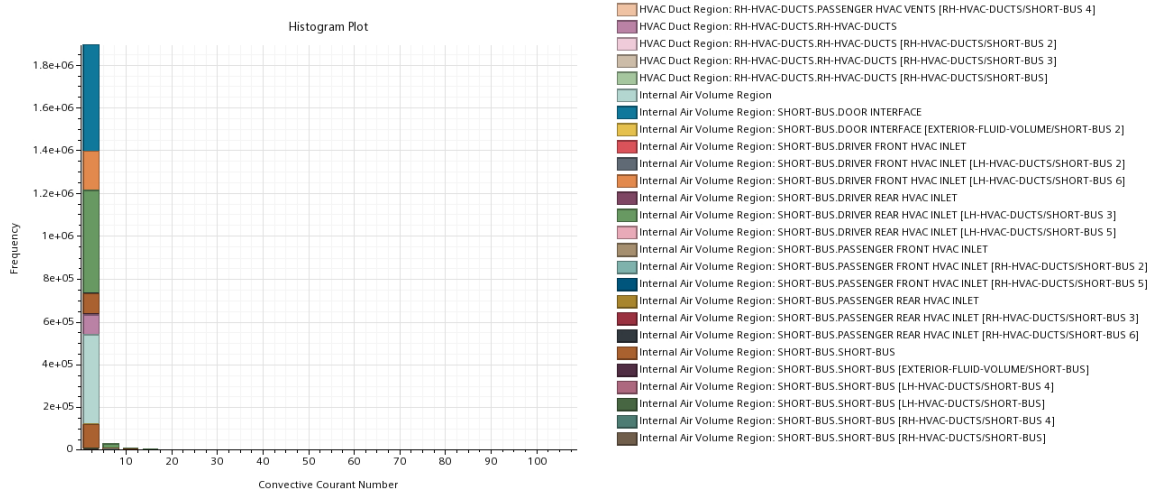


Figure 24: Base case - convective Courant number histogram.

For implicit solvers, the convective Courant number can be in the range of 10-100 locally, however with an overall mean value close to one is ideal [3]. Based on the base case histogram above, the convective Courant values are reasonable, as the majority are concentrated around one, with a few regions extending beyond 10.

4.3.10 Residuals

The residuals from the base case simulation are presented above in Figure 25.

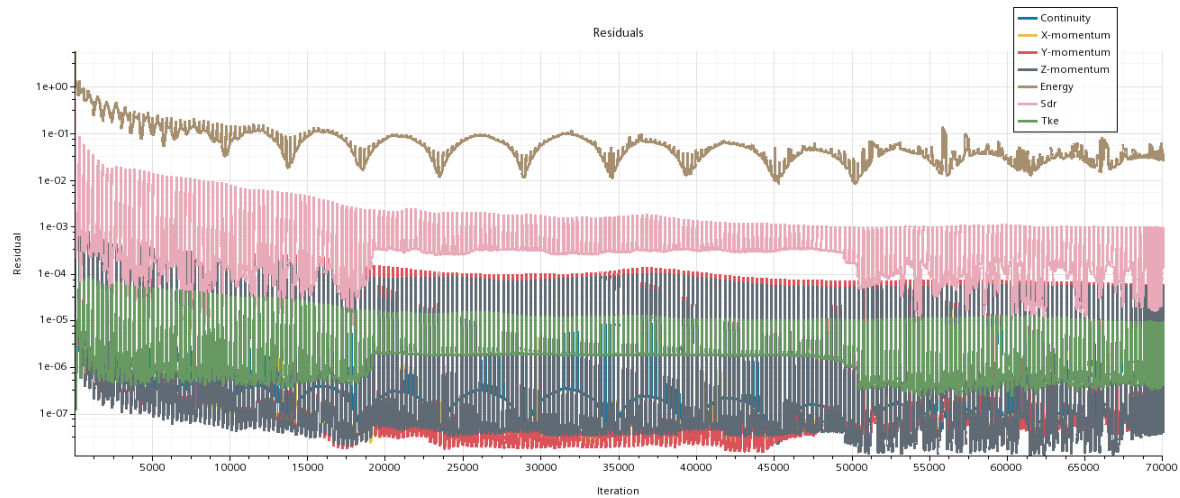


Figure 25: Base case - residuals.

The residuals represent the degree to which the discretized equations are satisfied [3]. The residuals contained in the monitor plot include, X-, Y-, and Z-momentum, continuity, energy, Tdr, and Tke. A general rule of thumb is the solution has converged when the residuals experience a drop greater than three orders of magnitude [3]. Although not a definite rule, as many factors can influence the residual drop including the initial guess [3].

4.4 Optimized Base Case CFD Analysis

Due to the extreme complexity and detail of the initial base case model, the solving time for an implicit unsteady 7 s transient simulation was over 90 hours. In order to increase the computational efficiency of the simulation, the following modifications were made to the CFD model.

1. Decreased the bus volume by $>65\%$.
2. Omitted the HVAC ducts from the volume mesh.
3. Removed mesh refinement in areas of unimportance.
4. Added door leafs as baffles to the model.

The above allowed for a significant reduction in solving time. The new optimized simulation solved the 7 s transient solution in 16 hours without automated scene capture running and 17 hours with automated scene capture activated. This allowed for faster iteration and trouble shooting when setting up the base case model. Additionally, the file sizes reduced by 66% in size compared to the original base case simulation. This improved the speed of file transfer when moving files between Linux on the server and our personal machine for more in depth post processing of results. The two CAD models for the original base case and optimized base case are presented in Figure 26a and Figure 26b respectively below.

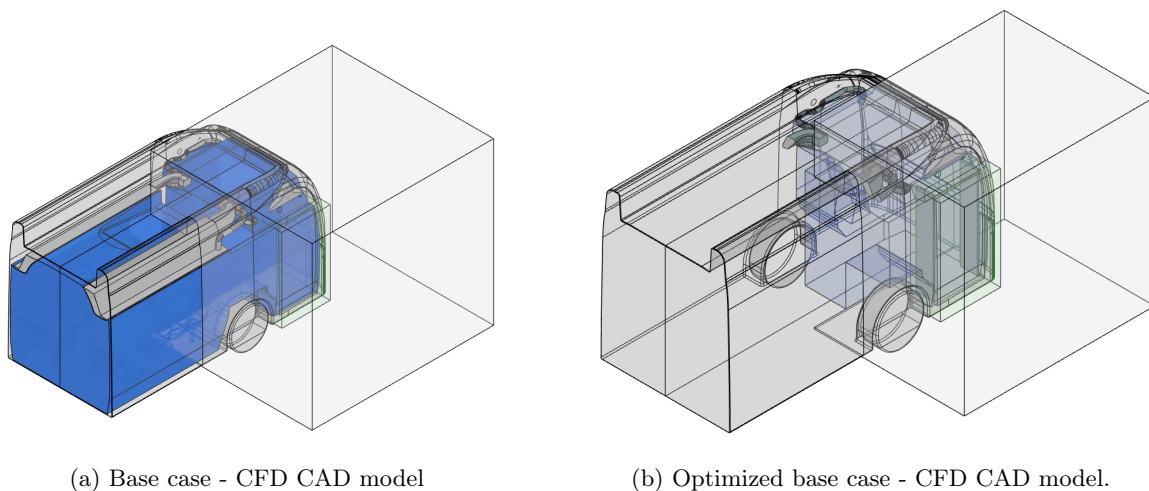


Figure 26: CFD CAD model comparison - fluid volume region for (a) base case and (b) optimized base case.

The large transparent box wrapping around the exterior of the bus represents the external fluid volume (ambient air) and the blue volume represents the internal fluid volume. The size of these regions directly influences the solving time of the simulation. As seen in the optimized base case, the internal volume is significantly smaller and the HVAC ducts have been removed from the mesh. These two modifications are the largest contributors to the increase in efficiency of the optimized base case simulation. A comparison of the mesh between the original base case and optimized base case is shown below in Figure 27.

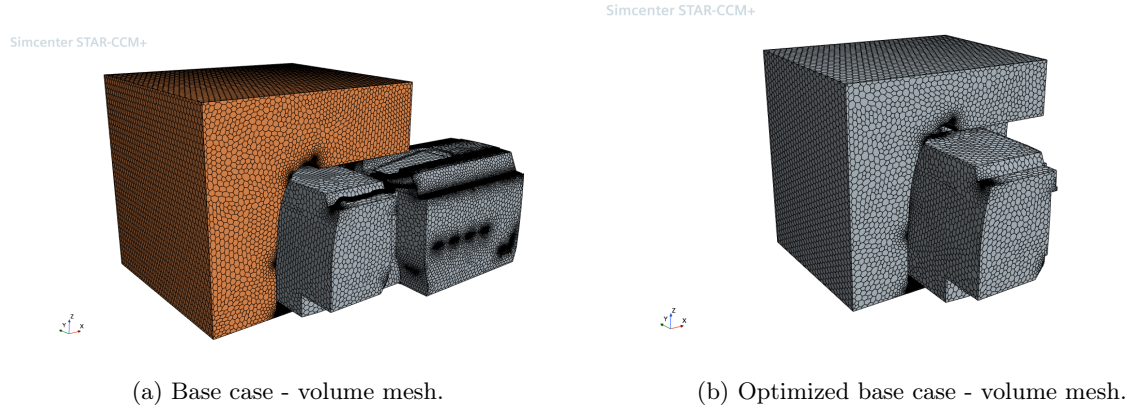


Figure 27: CFD mesh comparison - volume mesh for (a) base case and (b) optimized base case.

4.4.1 CFD Parameters

The CFD simulation parameters for the optimized base case are shown in Table XI below.

Table XI: OPTIMIZED BASE CASE - CFD SIMULATION INPUT PARAMETERS AND BOUNDARY CONDITIONS

Region	Boundaries/Interfaces	Boundary Condition	Value	Unit	Region Initial Temperature	Unit
External Air Region	Door Interface	Freestream	N/A	N/A	253	K
	Door Side Interface	Wall (No Slip)	N/A	N/A		
Internal Air Region	Driver Front HVAC Inlet	Mass Flow Inlet	0.1159	kg/s	288	K
	Driver Rear HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
	Passenger Front HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
	Passenger Rear HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
	Door Interface	Freestream	N/A	N/A		
	Internal Fluid Volume Walls	Wall (No Slip)	N/A	N/A		

NOTE: Refer to Section 4.3.1 for explanation of boundary conditions.

4.4.2 Velocity Streamlines

Figure 28 shows the velocity streamlines through the door boundary for the current New Flyer door system after 7 s of transient solution.

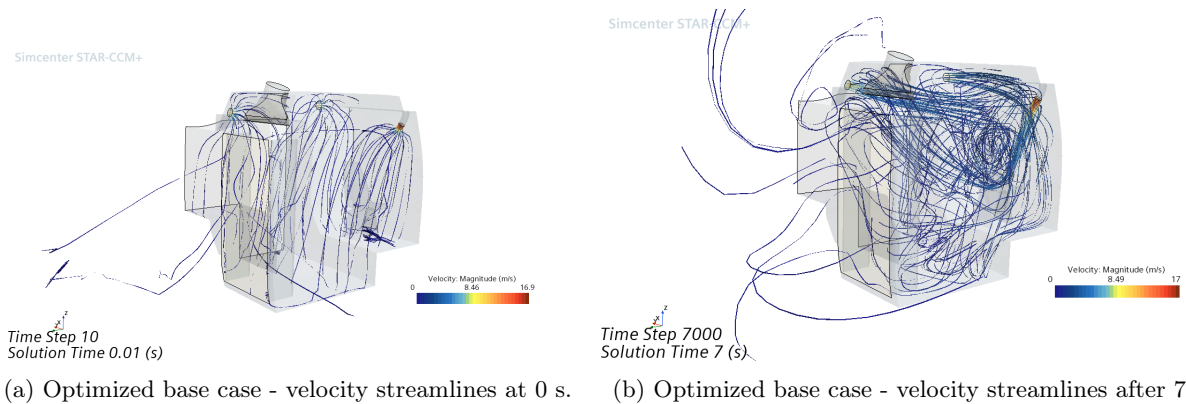


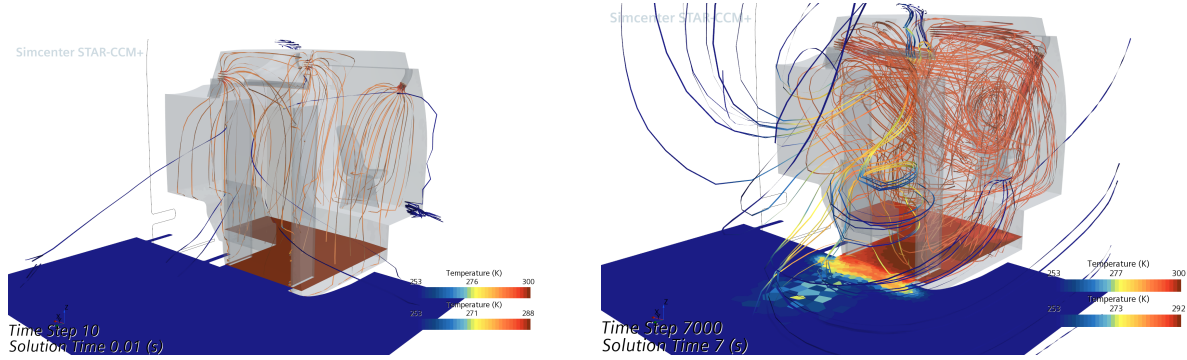
Figure 28: Base case - velocity streamlines through door boundary and propagation into external air medium for (a) 0 s and (b) 7 s of transient solution.

Comparing the 0 s solution to the developed 7 s solution, there is an increase in velocity streamlines recirculating and exiting through the bus door. This makes sense as the door has been open for a longer amount

of time. It should be noted that the CFD model does not include an HVAC air return, therefore the mass flow rate added to the bus is increasing the overall air volume inside the internal air volume. This further contributes to the increase in streamlines through the door boundary over time.

4.4.3 Temperature Gradient

Figure 29a shows the temperature gradient 1 s after the door opens and Figure 29b shows the temperature gradient after 7 s of transient solution respectively.



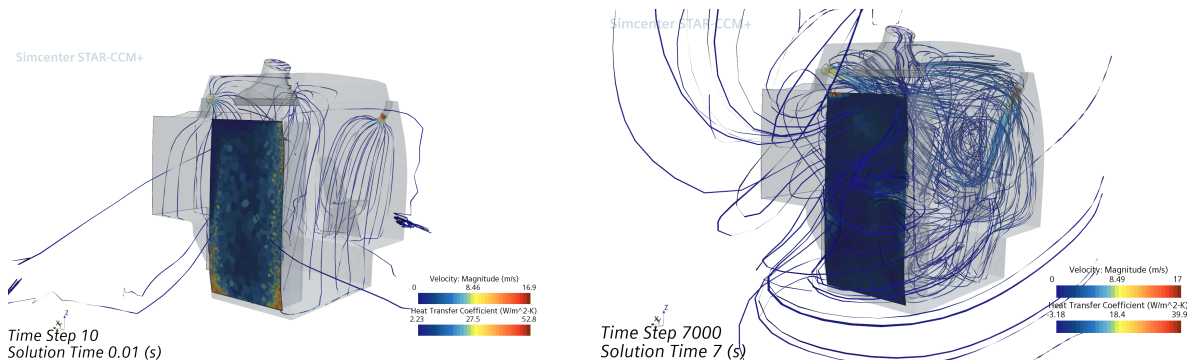
(a) Optimized base case - temperature gradient after 0 s. (b) Optimized base case - temperature gradient after 7 s.

Figure 29: Optimized base case - temperature gradients through door boundary and propagation into external air medium for (a) 0 s and (b) 7 s of transient solution.

After 7 s of transient solution, the air in the direct path of the passenger door exit has increased in temperature due to the mass flow of conditioned cabin air exiting the bus. This increase in temperature is shown on the section plane displaying the temperature gradient through the door boundary. The upper temperature scale in Figure 29 belongs to the velocity streamlines and the lower temperature scale corresponds to the section plane. Note the change in streamline temperature as it exits the door in Figure 29b.

4.4.4 Heat Transfer Coefficient

The heat transfer coefficient for the door boundary at 0 s and 7 s is presented in Figure 30a and 30b respectively.



(a) Optimized base case - heat transfer coefficient after 0 s. (b) Optimized base case - heat transfer coefficient after 7 s.

Figure 30: Optimized base case - heat transfer coefficient through door boundary for (a) 0 s and (b) 7 s of transient solution.

The boundary surface averaged heat transfer coefficient at 7 s of transient solution is $1.1538 \text{ W/m}^2\text{K}$. This

is significantly lower than the $1.781 \text{ W/m}^2\text{K}$ value from the original base case. This has been attributed to a simplification performed in the optimized base case model. The optimized base case did not include a region for the HVAC at a higher initial temperature, whereas the base case did. This is the probable cause for the increased heat transfer coefficient, boundary heat flux, and larger overall heat transfer in the original base case simulation.

4.4.5 Boundary Heat Flux

Figure 31 shows the boundary heat flux through the door boundary.

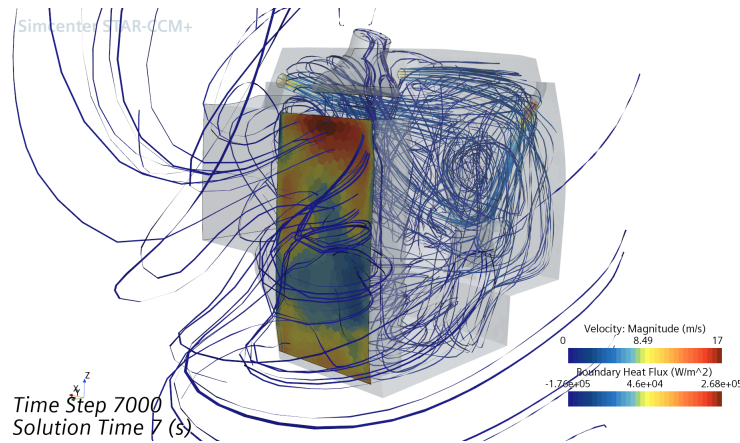


Figure 31: Optimized base case - boundary heat flux through door boundary.

The positive boundary heat flux indicates heat is transferred out of the region (i.e. out of the bus). The average surface value of boundary heat flux at 7 s of transient solution is $5.636 \times 10^4 \text{ W/m}^2$. Compared to the original base case which had a surface average boundary heat flux of $6.36 \times 10^4 \text{ W/m}^2$ at 7 s. The distribution is very similar compared to the original base case, however, the gradient is slightly different due to the reduction in internal volume which has slightly altered internal flow patterns.

4.4.6 Mass Flow Rate Through Door Boundary

The following plot shows the rate at which mass is flowing through the door boundary.

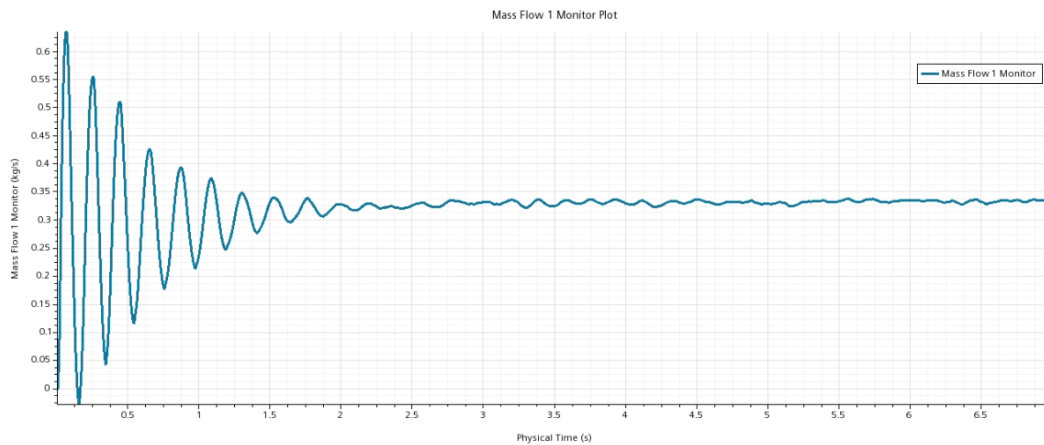


Figure 32: Optimized base case - transient mass flow rate through door boundary.

The transient mass flow rate converged to $\sim 0.325 \text{ kg/s}$ at approximately 2 s into the 7 s door cycle.

4.4.7 Heat Transfer Through Door Boundary

Figure 33 presents the transient heat transfer rate response over the 7 s door cycle.

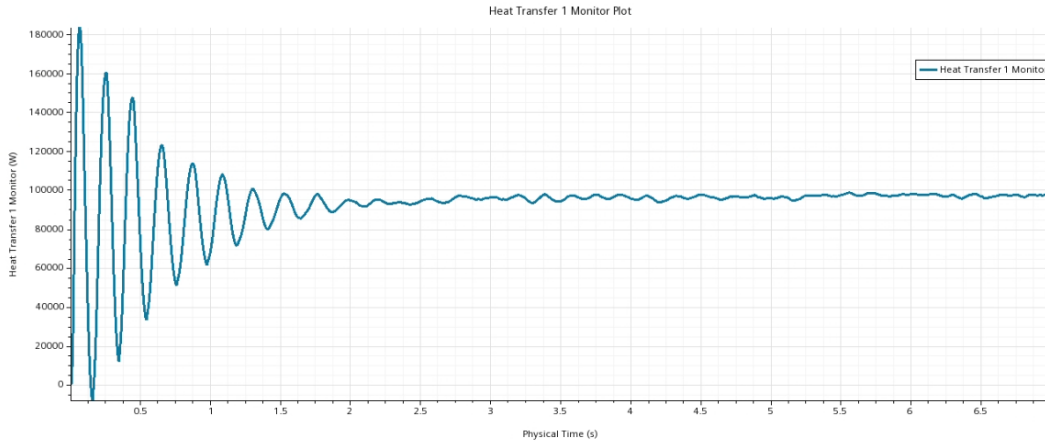


Figure 33: Optimized base case - transient heat transfer response through door boundary.

The value is oscillating, however appears to converge to ~ 97 kW of instantaneous heat transfer rate through the door boundary for loading periods greater than 7 s. This is 8 kW less than the value found in the original base case simulation. There are many simplifications to the volume for the simplified base case, although this seems to be a reasonable trade off of accuracy for computational efficiency

4.4.8 Wall Y+ Histogram

Figure 34 presents the Wall Y+ values for each boundary for the optimized base case simulation.

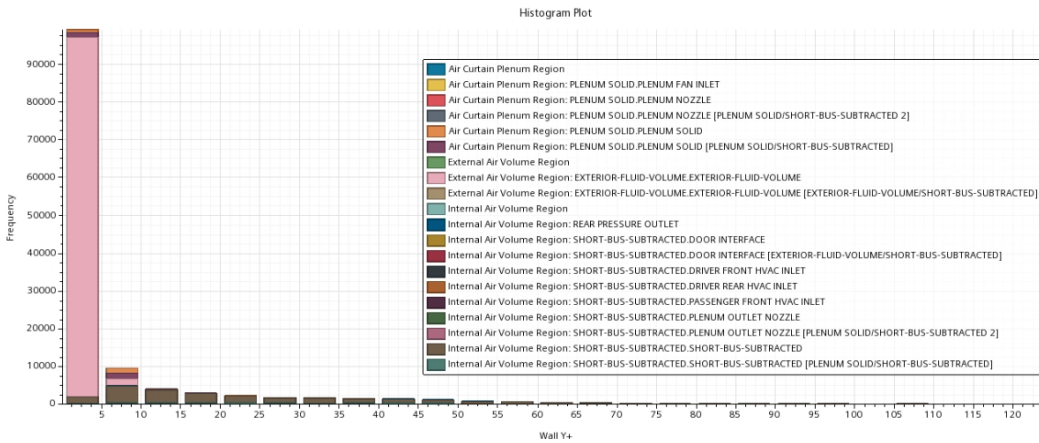


Figure 34: Optimized base case - wall Y+ histogram.

Due to the relatively low Reynolds number, the wall Y+ values for the optimized base case are reasonable as they do not surpass ~ 100 and on average are below 5.

4.4.9 Convective Courant Number Histogram

Figure 35 shows the convective Courant number for all of the regions and boundaries.

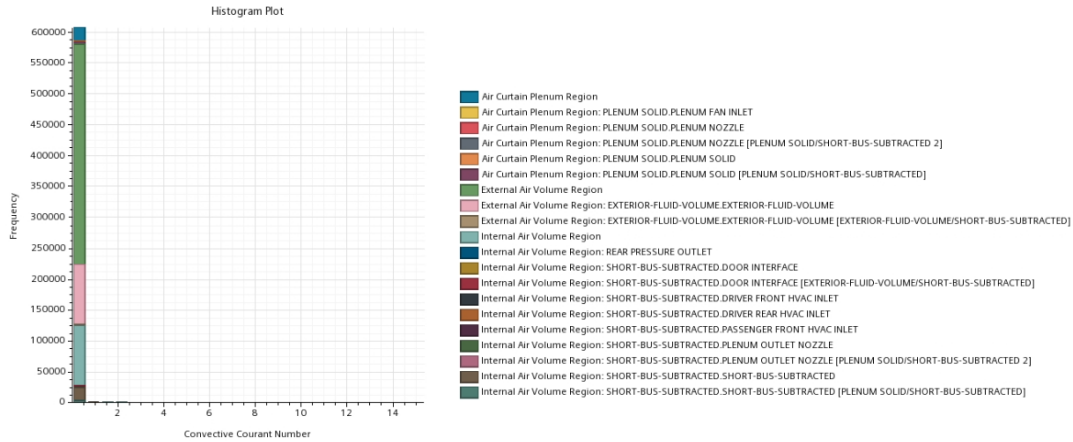


Figure 35: Optimized base case - convective Courant number histogram.

The convective Courant numbers for the optimized base case are very promising. It is ideal that this is close to one and based on the histogram, the majority of the boundaries fall in this range.

4.4.10 Residuals

The residuals from the optimized base case are presented above in Figure 36.

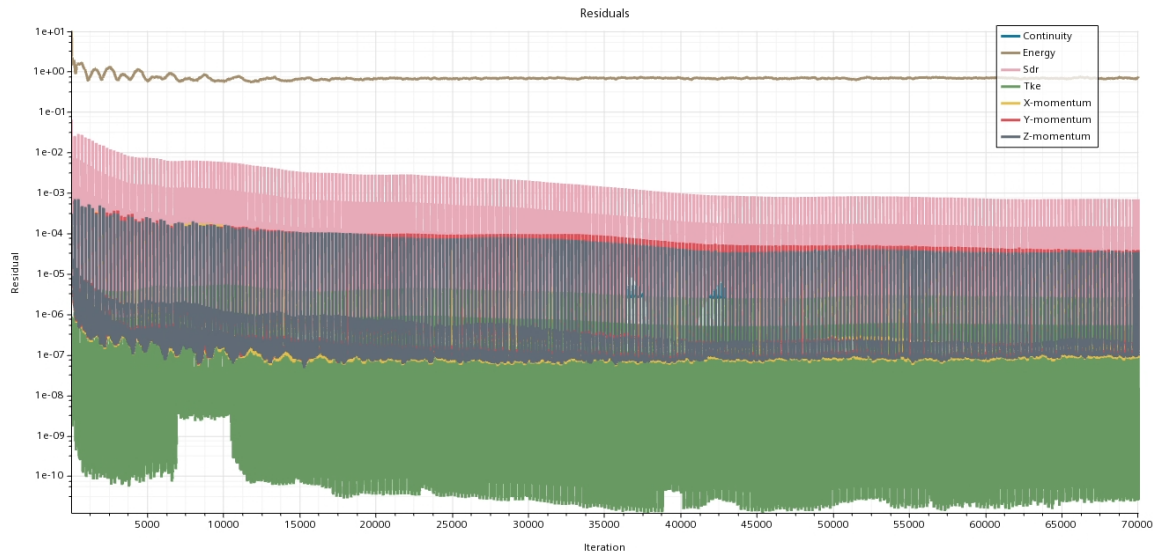


Figure 36: Optimized base case - residuals.

All residuals aside from energy experienced a minimum drop of three orders of magnitude, indicating a reasonably converged solution.

4.5 Initial Analysis of Air Curtain

Initial CFD analysis of the design with 860 CFM discharged from 32 in^2 nozzle exit, obtained through research showed that it was not strong enough to cover the full door opening and was deviating midway. Figure 37 shows the results for this case. It can be seen that the flow is discharging at a lower velocity from the nozzle.

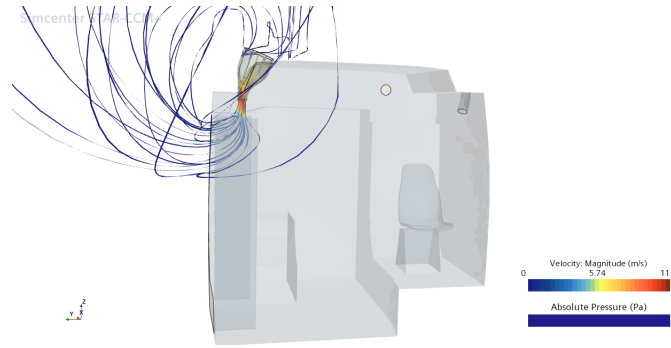


Figure 37: Initial analysis for 860 CFM.

The speed achieved in this case is ineffective in preventing heat transfer through the door. As a result, a higher velocity flow was required to form an air door. Analytically obtaining the flow speed is a very complex procedure as there are various changing factors that need to be considered at each time step to understand the flow pattern, and hence iterative CFD simulation studies were carried out with 3600, 2800, and 2200 CFM to determine the correct/range of velocity for the flow to effectively seal the door opening. Due to the complexity, time-consuming CFD analysis study, and limited time to complete the project, only the studies with the flow rates mentioned above were carried out. With 3600 CFM, a higher velocity was observed, the similar results were observed with 2800 CFM. The velocities were relatively high for the axial fan to produce. CFD study with 2200 CFM showed promising results that could be achieved with velocities of 23m/s when reaching the average passenger head height of 70 inches.

5 Final Design

The air curtain final design consists of a rotational molded polyethylene (PE) plenum that packages harmoniously with the existing New Flyer bus door mechanism. The final design is presented below in Figure 38.

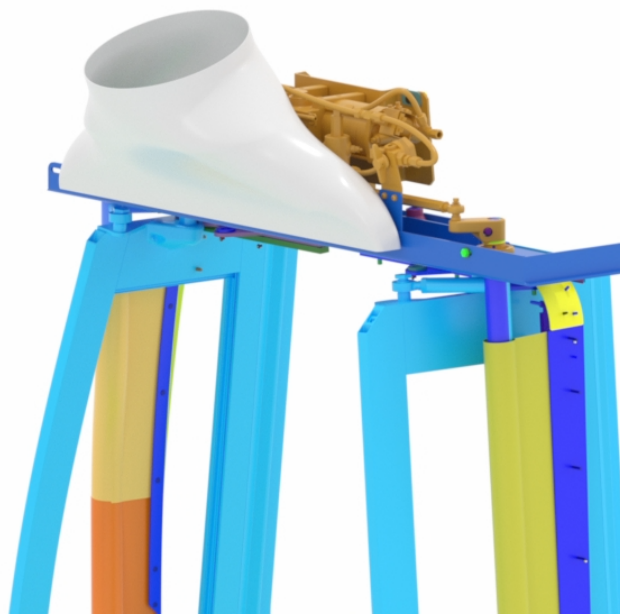
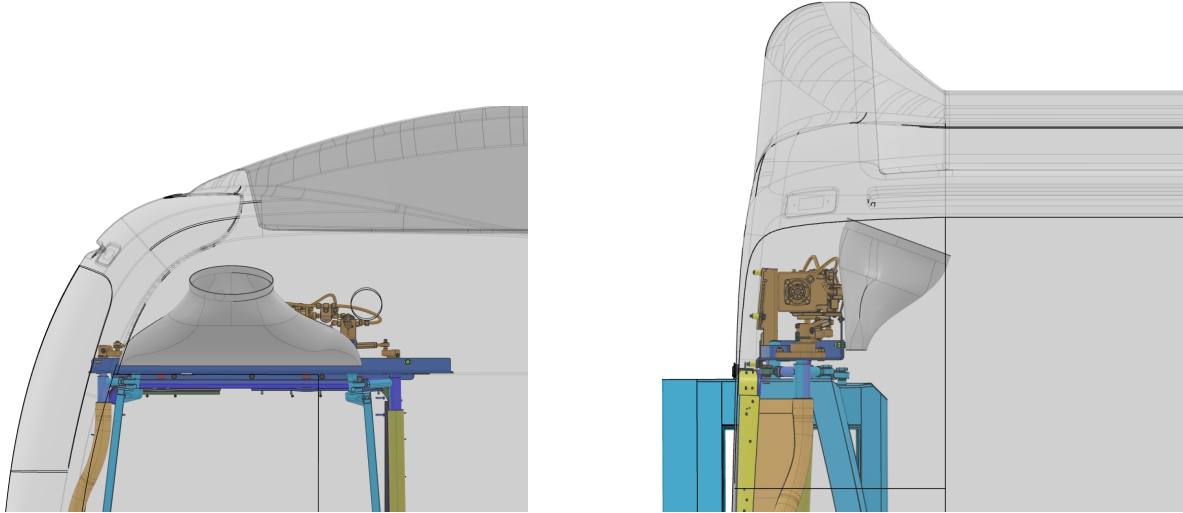


Figure 38: Final Design - air curtain with entrance door in open position.

The position of the plenum was decided with air curtain and packaging efficiency in mind. Additionally,

by maintaining the position of the existing door mechanism, the air curtain upgrade could be retrofitted to existing buses in the New Flyer test fleet, and if deemed successful, be available for sale as an official aftermarket item for existing New Flyer customers in regions with large temperature deltas. Figure 39 shows the air curtain in context of the New Flyer Bus A-class surface.



(a) Final Design - front view in context of the bus A-class surface. (b) Final Design - side view in context of the bus A-class surface.

Figure 39: Final Design - air curtain (a) front view and (b) side view in context of the bus A-class surface.

Detailed CFD analysis of the final design is found below in Section 5.1. Based on initial CFD of plenum geometry, the required CFM of the fan system was determined to be 2200 CFM to achieve the required nozzle velocity to adequately seal the door boundary. Twin centrifugal fans were spec'd to meet this volumetric flow rate demand. The fan selection is developed in Section 5.2.

5.1 Air Curtain CFD Analysis

The final air curtain analysis is presented below. Lots of trial and error went into determining the proper air parameters to optimize the results of the simulation. The simulation volume remained constant with the optimized base case simulation to provide an accurate comparison. That being said, a new subtract of the internal volume had to be performed in CATIA V6 to create the boundary surfaces of the air curtain plenum. The specified boundary condition parameters for the air curtain analysis are presented below in Table XII.

5.1.1 CFD Parameters

The CFD simulation parameters are shown in Table XII below.

Table XII: AIR CURTAIN - CFD SIMULATION INPUT PARAMETERS AND BOUNDARY CONDITIONS

Region	Boundaries/Interfaces	Boundary Condition	Value	Unit	Region Initial Temperature	Unit
External Air Region	Door Interface	Freestream	N/A	N/A	253	K
	Door Side Interface	Wall (No Slip)	N/A	N/A		
Air Curtain Plenum Region	Plenum Fan Inlet	Mass Flow Inlet	1.25	kg/s	288	K
	Plenum Nozzle Outlet	Pressure Outlet	10000	Pa		
Internal Air Region	Door Interface	Freestream	N/A	N/A	288	K
	Internal Fluid Volume Walls	Wall (No Slip)	N/A	N/A		
	Driver Front HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
	Driver Rear HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
	Passenger Front HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		
	Passenger Rear HVAC Inlet	Mass Flow Inlet	0.1159	kg/s		

The mass flow rate of 1.25 kg/s was used for the air curtain inlet as that corresponds to (~2200 CFM). Slower mass flow rates corresponding to 860 CFM were trialed initially, however did not reach the required nozzle outlet velocities due to our plenum nozzle geometry. Had the project had more time or resources, further optimization of the plenum geometry could have been performed in order to solve with a lower flow rate. A pressure outlet of 10,000 Pa was specified as the boundary condition of the plenum nozzle from the pressure drop observed in the steady-state plenum only simulation that was performed. The mass flow rates of HVAC inlets remained identical from the base case and optimized base case simulations at 0.1159 kg/s. Additionally, the region temperatures remained constant between this simulation and the optimized base case. The air curtain was assumed to be drawing air in from inside the bus, therefore the temperature of the air leaving the air curtain was specified to be at 288 K. As discussed in the mass flow rate and heat transfer monitor plots in Section 5.1.8, this resulted in significant losses due to the air curtain air crossing the boundary.

5.1.2 Velocity Streamlines

The velocity streamlines for the air curtain at 1 s and 7 s is presented in Figure 40a and 40b respectively.

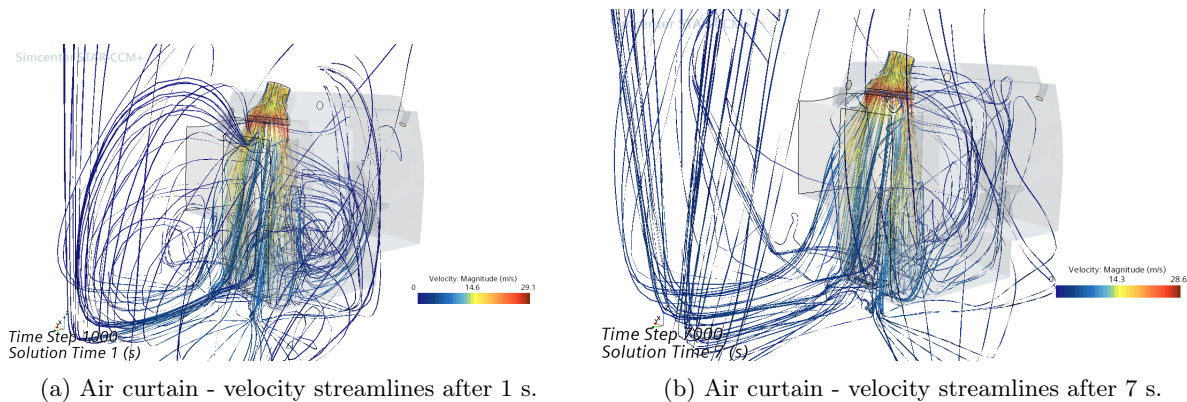


Figure 40: Air curtain - velocity streamlines through door boundary and propagation into external air medium for (a) 1 s and (b) 7 s of transient solution.

Figure 40 shows the velocity streamlines through the door boundary for the current New Flyer door system after 1 s and 7 s of transient solution respectively. The two figures show the progression of the air curtain angle to increase with respect to the bus. This results in more air being pushed outside of the bus. Since heat transfer is being calculated by tracking the mass flow through the door boundary, the air exiting the door from the air curtain plenum is causing significant losses. There is a certain error introduced that worsens as time progresses in the transient simulation due to the lack of HVAC air return, effectively causing the internal pressure to increase, resulting in the air curtain performing worse as time goes on.

An HVAC return was in development, although was discontinued for use in this project due to the limited time and resources remaining to complete the project and satisfy all stakeholders. The vertical streamlines in the external volume region are due to the flow interacting with the walls of this region. The external air volume could not be extremely large, due to the immense computing requirements of the simulation, however, this must be taken into account when examining the results.

5.1.3 Temperature Gradient

Figure 41a shows the temperature gradient 1 s after the door opens and Figure 41b shows the temperature gradient after 7 s of transient solution respectively.

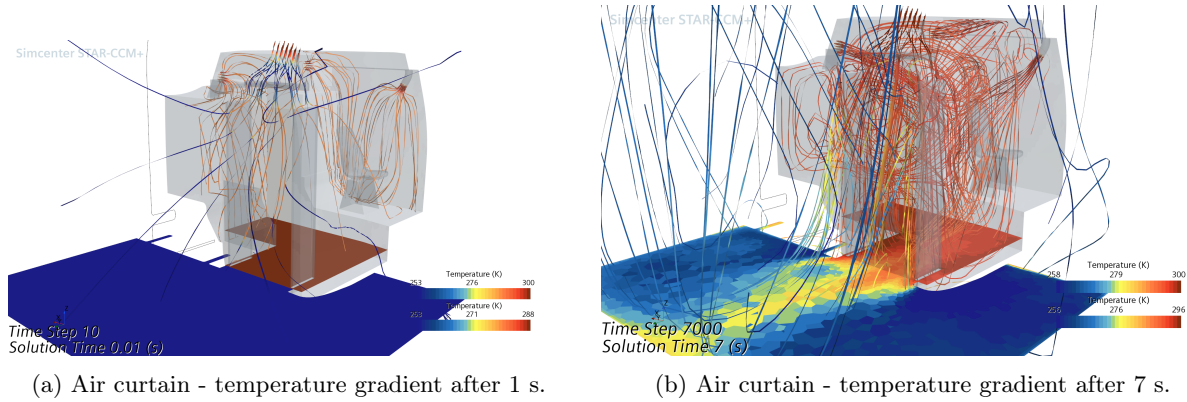


Figure 41: Air curtain - temperature gradients through door boundary and propagation into external air medium for (a) 1 s and (b) 7 s of transient solution.

Similarly to the optimized base case, the temperature in the external fluid region directly outside of the door boundary increases in temperature as the time step increases. Comparing the air curtain (Figure 41b) versus the optimized base case (Figure 29b), it is clear that the air curtain is actually causing more hot air to exit the bus according to the simulation. This is largely due to the fact that the air coming from the air curtain is flowing out of the bus after contacting the floor of the bus rather than recirculating inside the bus. Since the heat transfer across the boundary is calculated using the mass flow rate crossing the boundary, this results in higher losses when compared to the base case and optimized base case simulations. Future iterations with varying nozzle angles with respect to the Z-coordinate direction are required to improve this condition. There is a serious amount of time investment in order to parameterize the CFD simulation which the team did not have time to undertake in the short span of this project. Additionally, setting up multiple simulations from scratch with varying angles would have been too time consuming and would not allow any time for the interpretation and presentation of final results. For these reasons, these tests have been omitted from this report. The team had to balance the requirements and use all of the available bandwidth to ensure the project was completed on time. Further development of this topic is included in Section 6.

5.1.4 Heat Transfer Coefficient

The heat transfer coefficient for the door boundary at 7 s is presented in Figure 42.

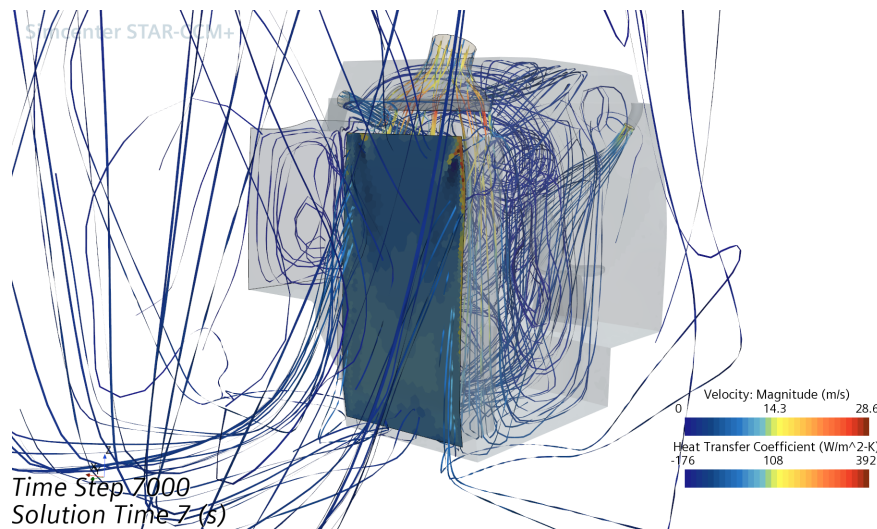


Figure 42: Air curtain - heat transfer coefficient through door boundary.

At 7 s, the heat transfer coefficient surface average at the door boundary is $15.06 \text{ W/m}^2\text{K}$. This is significantly higher than the base case and optimized base case result. Again, this is due to the air exiting the air curtain plenum that is at the internal bus temperature that results in the significant increase in heat transfer coefficient, boundary heat flux, mass flow rate across the boundary and the increased overall heat transfer.

5.1.5 Boundary Heat Flux

The boundary heat flux through the door boundary at 7 s is presented in Figure 43 below.

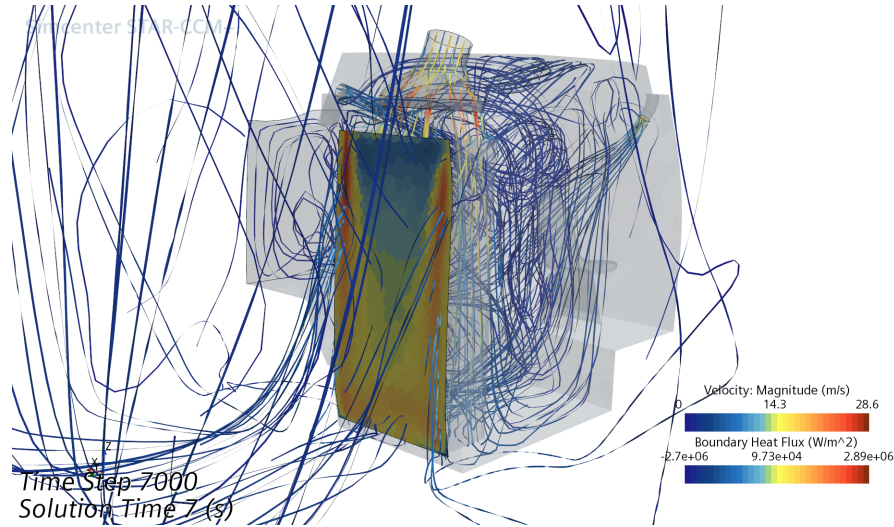
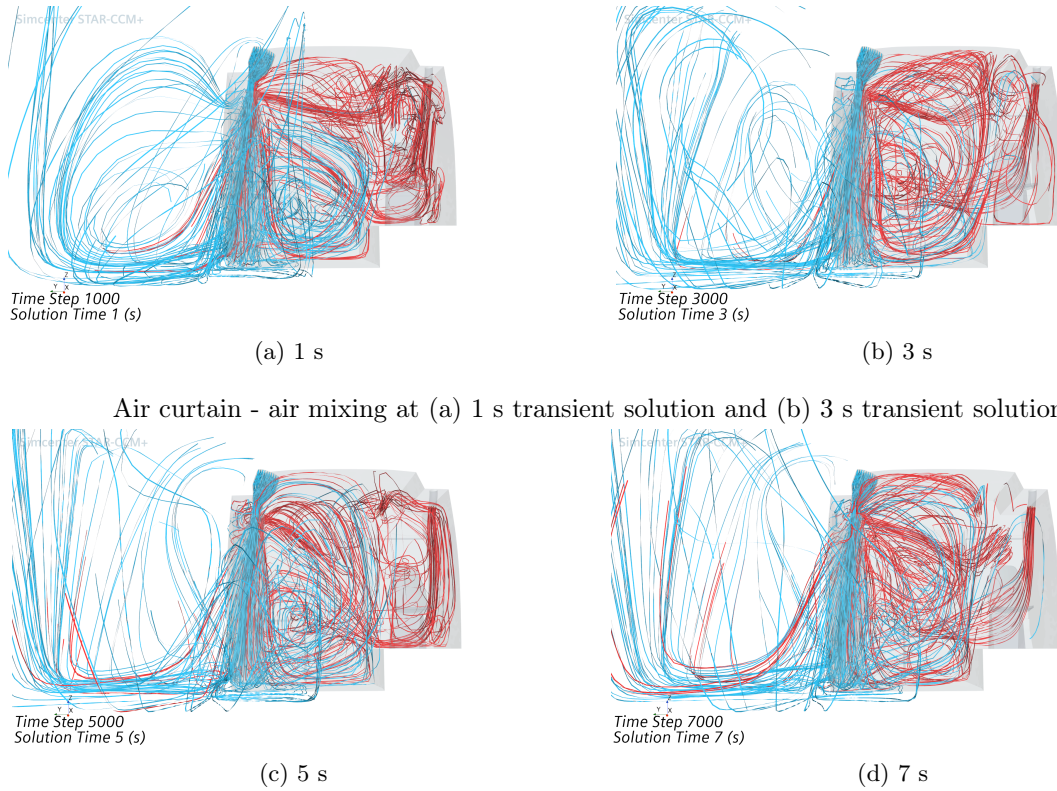


Figure 43: Air curtain - boundary heat flux through door boundary.

The average surface value of boundary heat flux for the air curtain simulation at 7 s of transient solution is $2.217 \times 10^5 \text{ W/m}^2$. The boundary heat flux for the air curtain case is noticeably larger compared to the optimized base case and the original base case which had a surface average boundary heat flux of $6.36 \times 10^4 \text{ W/m}^2$, and $5.636 \times 10^4 \text{ W/m}^2$ respectively at 7 s.

5.1.6 Air Mixing

Another way to visualize the effectiveness of the air curtain is to look at the interaction of the internal conditioned air and the air curtain air. It is important for the air curtain to contain this conditioned air in the cabin and reduce the amount that is able to cross through the door boundary. Figure 44 below shows the mixing of the air curtain air and the internal cabin air throughout a 7 s door cycle.



Air curtain - air mixing at (a) 1 s transient solution and (b) 3 s transient solution.

Air curtain - air mixing at (c) 5 s transient solution and (d) 7 s transient solution.

Figure 44: Air curtain - air mixing throughout a 7 s door cycle.

Red streamlines: The conditioned air exiting the internal HVAC outlets at the internal temperature of 288 K.
 Blue streamlines: The air exiting the air curtain plenum at the same temperature of 288 K.

Figure 44, shows that the air curtain is promoting the internal air to recirculate in the internal bus region instead of existing through the bus door. As the time step approaches 7 s, the amount of conditioned red air exiting the door is increasing. This is largely due to the fact that this simplified model did not contain an HVAC air return or pressure vent of any kind. This means that the added mass flow rate of $4 \times 0.1159(kg/s) = 0.4636(kg/s)$ is unable to vent from any external pressure release. This is due to the fact that the bus internal volume was shrunk in order to improve the computational efficiency and the bus is entirely manifold (meaning there are no gaps for the air to escape or release the built up pressure.) This is observed in Figure 44 as the air curtain air is forced out of the bus at an increasing angle as the time step increases. This presents some inaccuracy in the model and should be considered when evaluating the results. Buses naturally do not have the most demanding air tightness requirements, and this along with air returns would negate this pressure build up experienced in this simulation.

5.1.7 Mass Flow Rate Through Door Boundary

Figure 45, shows the rate at which mass is flowing through the door boundary for the air curtain simulation.

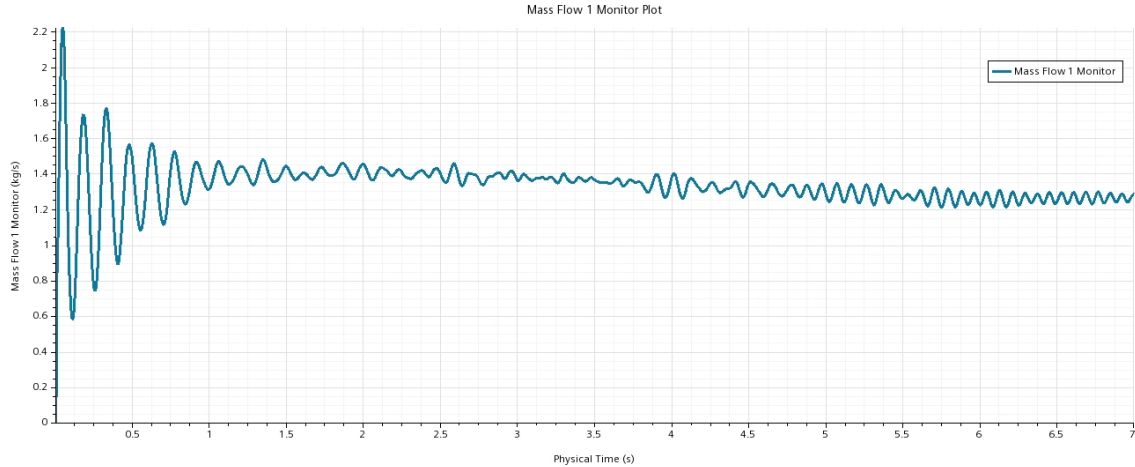


Figure 45: Air curtain - transient mass flow rate through door boundary.

The transient mass flow rate converged to ~ 1.3 kg/s at approximately 4.5 s into the door cycle compared to ~ 0.325 kg/s for the optimized base case results. This increase in mass flow rate of conditioned air passing through the boundary increased the yielded heat transfer of the system.

5.1.8 Heat Transfer Through Door Boundary

Figure 46 presents the transient heat transfer response over the 7 s door cycle for the air curtain case.

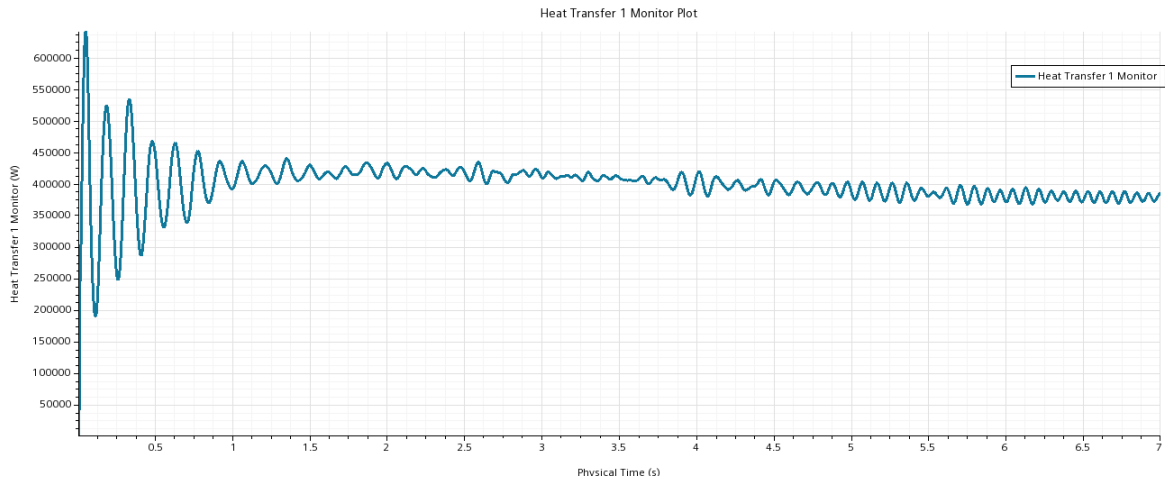


Figure 46: Air curtain - transient heat transfer response through door boundary.

The value is oscillating, however appears to converge to ~ 380 kW of instantaneous heat transfer rate through the door boundary for loading periods greater than 7 s. That is significantly more heat transfer compared to the ~ 97 kW of heat transfer rate calculated in the optimized base case simulation. To reiterate, this is due to the internal temperature air leaving the air curtain plenum outlet that is crossing the door boundary. This value could absolutely be improved through various parameters including nozzle size, nozzle angle, nozzle proximity from the door etc. Additionally, a lower air curtain return to promote re-circulation into the bus cabin could drastically improve this result.

5.1.9 Wall Y+ Histogram

Figure 47 presents the Wall Y+ values for each boundary for the air curtain simulation.

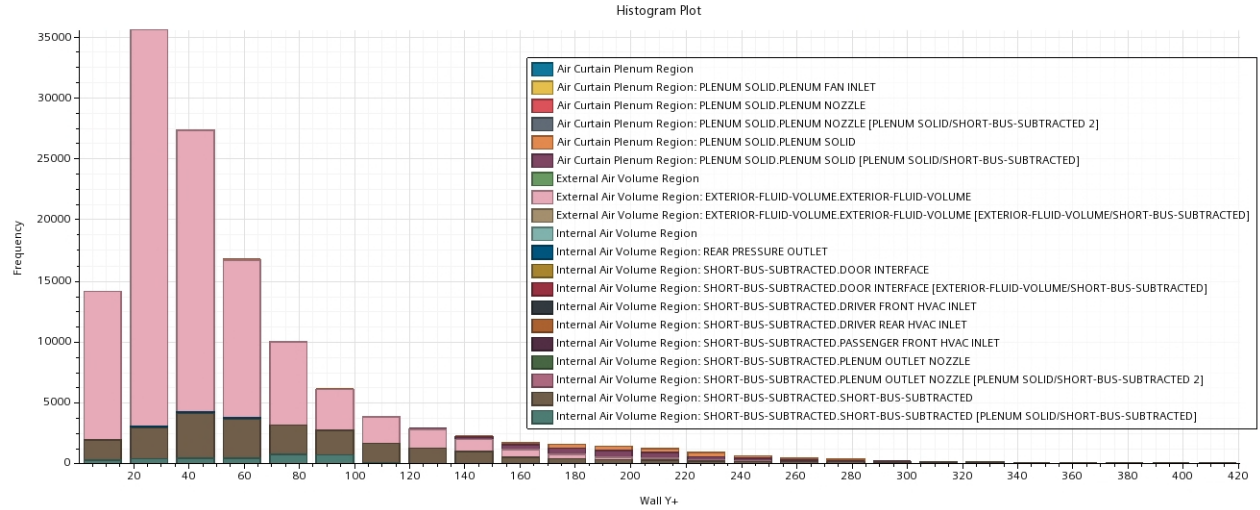


Figure 47: Air curtain - wall Y+ histogram.

Although the Y+ values are higher compared to the base case and optimized base case, this is expected due to the higher velocity flow and thus Reynolds number. Therefore there is a higher tolerance of increased wall Y+ values for the air curtain simulation compared to the base case simulations.

5.1.10 Convective Courant Number Histogram

Figure 48 shows the convective Courant number for all of the regions and boundaries.

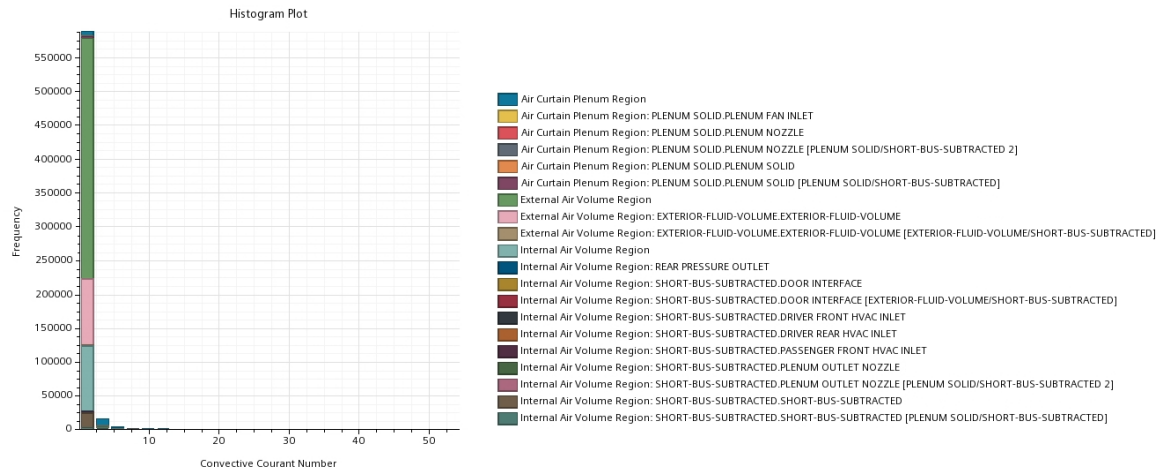


Figure 48: Air curtain - convective Courant number (CFL) histogram.

The convective Courant number (CFL) for the air curtain are reasonable as the mean Courant number is 1. The convective Courant number controls the size of the local time step in an implicit unsteady simulation. This CFL number serves as a relaxation factor and indicates that the time step used of 0.001 s is providing adequate results [3].

5.1.11 Residuals

The residuals for the air curtain simulation are presented above in Figure 49.

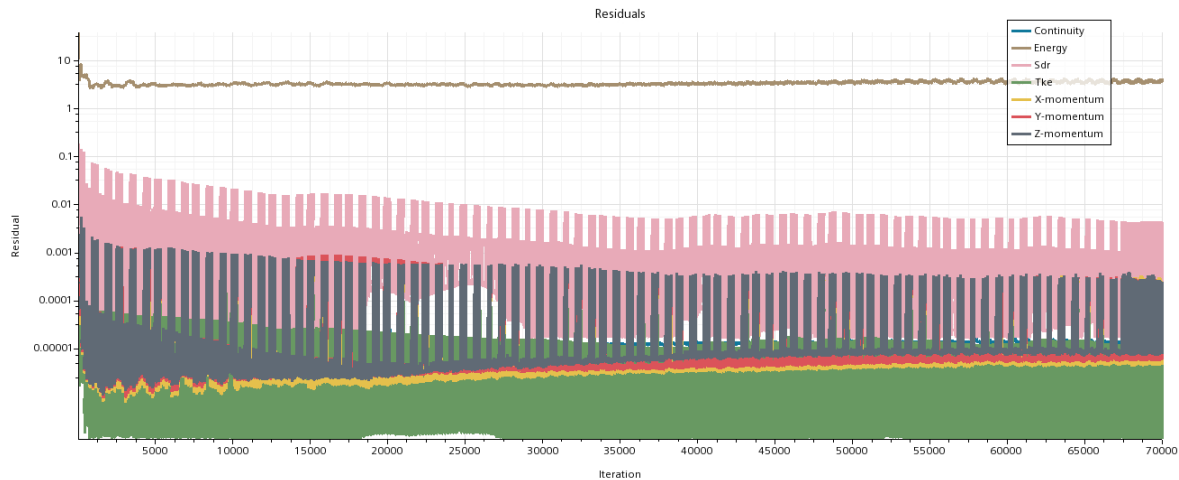


Figure 49: Air curtain - residuals.

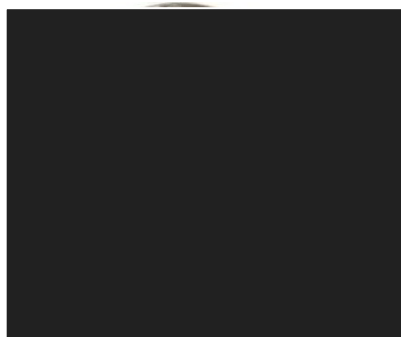
The residuals for the air curtain simulation have experienced a similar three order of magnitude drop aside from the energy residual. This generally indicates a converged solution. For the purposes of our analysis, each simulation performed showed adequate convergence.

NOTE: Review Section 4.3.10 for a brief explanation of the residual terms and general convergence.

5.2 Fan

A fan was selected to meet the required CFM of 2200 as indicated in CFD analysis. Fans that already exist were analyzed for use in the air curtain as this part would be a purchased part from a vendor. A single axial fan, powered by 24 VDC does not meet the requirements for this application. The formation of back pressure during the fan operation in the duct meant that a higher CFM-producing fan needed to be sourced.

The team decided to switch from an axial fan to a centrifugal fan to meet the airflow requirements that are higher than previously anticipated. However, since the centrifugal fan is limited to producing lower CFM, a dual centrifugal fan configuration is used. A dual fan would generate the required CFM with better laminar flow. Figure 50 shows the fan selected for the air curtain. The fan operates on 24 VDC with a 175.2W power rating. The data sheet provided in Appendix B.2 outlines the specifications for the fan.

Figure 50: Centrifugal fan for the final design.
[18]

Analysis on the fan was carried out to determine the energy consumption and range loss for the bus due to fan operation. The analysis was carried out for one trip of Winnipeg transit St.Norbert Blue route as this route simulates a route that an Xcelsior Charge bus equipped with an air curtain would be used on.

To determine the power consumption of the fan, it was important to determine the number of times the fan would be used during a trip. Since the fan would only be in operation when the door opened, a number of stops on the route were needed. According to Winnipeg Transit, St.Norbert Blue route has 47 stops. Therefore, the fan would be in operation 47 times during this trip [15]. The operation time for the fan is assumed to be nine seconds as the fan will be switched on before the door mechanism is activated and switched off when the door mechanism is deactivated. Energy consumption and range loss due to the fans for a single trip and for a 12-hour bus operation are calculated as:

The energy consumption of the fans for the trip is found by:

$$E_{Fan} = P_{fan} * t * N_{stops} \quad (10)$$

Where E_{Fan} is the fans energy consumption, P_{fan} is the fan operating power, t is the door operation time and N_{stops} is the number of times the door opens.

$$E_{Fan} = 0.3504kW * (\frac{9}{3600})h * 47 = 0.04117kWh$$

For a single trip for the bus, 0.04117 kWh of energy is consumed by the fans. Given the battery capacity of 446.25kWh and range of 402Km for the bus mentioned in section ??, the range loss per trip due to fans was found to be:

$$R_{Loss/trip} = \frac{Range}{BatteryCapacity} * E_{Fan} \quad (11)$$

$$R_{Loss/trip} = \frac{402(km)}{446.25(kWh)} * 0.04117(kWh) = 0.03711km/trip$$

For a single trip, the bus losses about 37.12 m of the range.

The battery capacity $B_{consumption}$ used by the fans for a single trip is calculated as:

$$B_{consumption} = \frac{E_{Fan}}{B_{capacity}} * 100 \quad (12)$$

$$B_{consumption} = \frac{0.04117(kWh)}{446.25(kWh)} * 100 = 0.00923\%$$

$$B_{consumption} = 0.00923\%$$

Taking the operational time of 12h for the bus in a day and averaging the time of 40 min for the completion of one trip, the bus is able to make 18 trips (N_{Trips}) in 12 hour time period.

Therefore, the range loss (R_{Loss}) in the 12h time period is calculated with Equation 13:

$$R_{Loss} = R_{Loss/trip} * N_{Trips} \quad (13)$$

$$R_{Loss} = 0.03711(km/trip) * 18 = 0.66802km$$

$$B_{consumption} = B_{consumption/trip} * N_{Trips} \quad (14)$$

$$B_{consumption} = 0.00923\% * 18 = 0.1667\%$$

$$B_{consumption} = 0.1667\%$$

The energy consumption by the fans for 12-hour operation is found using Equation 15:

$$E_{Fan} = E_{Fan/trip} * N_{Trips} \quad (15)$$

$$E_{Fan} = 0.04117(kWh) * 18 = 0.74110(kWh)$$

The bus loses close to 668m of range, with 0.1667% of battery usage for 12 hours of operation in a day due to the air curtain with 0.74110 kWh of energy consumption. This is significantly lower than the range loss due to the HVAC system used for heating and cooling the bus cabinet. Therefore, using this fan would be beneficial for the bus.

Table XIII summarizes the energy usage and range loss due to the fans for St.Norbert Blue route for 12 hour operation period.

Table XIII: FAN ENERGY USE SUMMARY FOR 12-HOUR OPERATION

Fan operation	
Range Loss (km)	0.66802
Battery Capacity Used (%)	0.16607
Energy Consumption (kWh)	0.74110

Based on the data presented in this table it is evident that the power requirement of the fan would result in a range loss that is only a fraction of the range loss due to heating losses. Thus, the return on this energy investment will be net positive if the air deposited by the air curtain can be retained in the bus. This relates to the main objective of the project which indicates that the project can be worthwhile to implement.

5.3 Control System

Regulating the use of the air curtain through a properly implemented control system will increase the efficiency of the bus. The control system would actuate the air curtain when the door mechanism is triggered and then stop the fans powering the air curtain when the door fully closes.

Another factor to consider in the control system design is the temperature ranges that it will operate in. When the ambient temperature is close to the set point temperature of the bus powering the air curtain would have increasingly diminishing returns. Thus, using further CFD analysis a data-set could be compiled to determine the temperature range where powering the air curtain is no longer an efficient use of power. Using temperature sensors that already exist on the bus the controller could be set up to automatically turn the air curtain system as a whole on or off.

A magnetic door contact control system can be used to regulate the air curtain. The air curtain switches on when the connection is deactivated by the opening action of the door and switches off the air curtain when the connection is complete, as the door is closed. A mechanical door contact can also be used for a similar purpose. Figure 51 shows the door contact control system that could be used with the air curtain. Integrating the control system with the PID controller would provide better control over the operation of the air curtain. Introducing a step delay in the controller allows the air curtain to be switched on before the door mechanism is activated. This will create a barrier before the door is opened and switches off when the door is completely closed, hence reducing the cabin air from exiting the bus.



Figure 51: Magnetic and mechanical door contact system.

[19]

5.4 Cost Analysis

The primary process for producing the part was selected to be rotational molding. High-density polyethylene (HDPE) resin was chosen for producing the part as HDPE is commonly used in rotational molding, has a high strength-to-density ratio, and is commonly recyclable. A roto-molding manufacturer was contacted to obtain the quote for produce the part. Shanghai Francom Rotomolding Technology Co., Ltd was contacted to receive their manufacturing charges. The charges quoted by the company are outlined in Table XIV.

Table XIV: ROTO-MOLDING CHARGES

Company Roto-molding Charges		
Weight (Kg)	Charges (USD)	MOQ
> 3	5.3/kg	100
< 3	15-19	100

The charges provided by the manufacturer are only applicable if more than 100 pieces are ordered. If fewer than 100 quantities are ordered, the prices increase. If the part was purchased from the manufacturer, the client will have to bear the charges for shipping transportation as these prices are marked with Ex works (EXW) by the manufacturer. The production of the plenum was quoted to be \$18 USD, while the mold price was quoted to be \$1950 USD. The quote from the manufacturer is shown in Appendix B.2 and B.1, respectively.

Detailed cost analysis and cost breakdown were not a high priority requirement as installation and installation procedures would change to incorporate NF standards. Hence, a detailed analysis of cost in terms of mounting and cost breakdown was not conducted. A prototype model would be required to carry out the proof-of-concept testing, hence rework instruction on the existing bus part for the air curtain, prototype CAD modeling, and installation of the air curtain are the main costs that will be incurred.

These costs are divided into two portions, mechanical cost, and electrical cost. To perform all the actions mentioned above, 40 hours of mechanical work and 16 hours of electrical work would be needed. Hourly charges of \$80 for engineering cost was discussed with the client. A detailed production can then be carried out on a prototype showing the desired results, which would cost an estimated 160 hours of mechanical work and 40 hours of electrical work. Table XV and Table XVI summarize the total cost for implementing the prototype and production-ready design into the bus, respectively.

Table XV: COST FOR PROTOTYPE DESIGN

Prototype work cost			
Work	Hours	hourly cost	Total Cost(CAD)
Mechanical	40	\$80	\$3200
Electrical	16	\$80	\$1280
Total			\$4480

Table XVI: COST FOR DETAILED PRODUCTION

Detailed production			
Work	Hours	hourly cost	Total Cost
Mechanical	160	\$80	\$12800
Electrical	40	\$80	\$3200
Total			\$16000

5.5 Failure Modes & Effects Analysis (FMEA)

This section determines the possible failure modes that can potentially occur at different stages of the air curtain development process using the FMEA rubrics provided in the Appendix C. Most risks were identified in the designing stage and were assigned a Risk Priority Number (RPN).

Table XVII: FMEA OF AIR CURTAIN

Items	Potential Failure Mode	Potential Effect	SEV	Potential cause	OCC	Current Control	DET	RPN
Designing	Interference of the air curtain model with existing bus parts	Air curtain model do not fit in the available space in the bus model	3	Not measuring the available space around the door mechanism correctly	6	Fitment	2	36
	Air curtain model fits in too tightly in the bus model	Tolerances are off during design CAD model	2	Lack of Inadequate measurements	5	Fitment	2	20
	Fan not fitting in the plenum	Remodeling the plenum with correct fan dimensions	3	Fan dimensions incorrectly measured	5	Fitment	1	15
	Fan do not produce required CFM	Not enough speed of airflow jet allowing the exchange of air through the airflow from the air curtain	7	Lack of proper Research/CFD Analysis of fan	2	Visual	3	42
	Plenum material fails to hold the airflow pressure from the fan	Improper material used to produce the plenum	7	Lack of proper stress analysis	2	Visual	2	28
	Jet of airflow produced by air curtain not fully laminar	Allows the air exchange through the airflow	8	Lack of proper CFD analysis of the fan with the plenum or inadequate plenum design to allow the airflow to develop into laminar	2	Visual	3	48
Engineering Drawings	Final plenum drawings are missing critical dimension	Drawing returned by manufacturer for correction/delayed production	3	Not carefully reviewing the final drawing before releasing it to the production	3	Visual	2	18
Forming Plenum	Plenum molded incorrectly	Re-production of the plenum/Delayed production	4	Not following the drawings carefully	2	Visual	3	24
Assembling air curtain	Wrongly assembled the air curtain	Delayed production	2	Not following the assembling instructions on the drawing from design team	3	Visual	3	18

Three failure modes with the highest risk were anticipated to occur during the design process of the air curtain. Table XVIII shows the corrective actions that can be taken to eliminate each of these failure modes.

Table XVIII: RPN OF AIR CURTAIN

RPN	Potential Failure Mode	Recommended Actions
36	Interference of the air curtain model with existing bus parts	Have the final model double checked for interferences
42	Fan selected do not produce required CFM	Carry out a proper CFD analysis of the air curtain before finalizing the fan to use
48	Jet of airflow produced by air curtain not fully laminar	Design a long plenum that will allow the airflow to develop into laminar flow/carryout CFD Analysis to further confirm that flow develops into laminar flow

6 Future Recommendations

In the interest of completing the project on time the design and corresponding analysis for solving the overlying problem of reducing heat loss through the door had to be restricted. Although only a limited design and analysis was completed, the air curtain solution proved to be an effective solution to the problem. Throughout the design process several ideas that could be used in future phases of the implementation of the air curtain were explored. The following sections outline several future recommendations for further analysis of the air curtain design and additional ideas to minimize energy loss through the door. These future recommendations are summarized below:

1. Future air curtain analysis

- (a) Airflow during passenger ingress and egress
- (b) Address pressure build up in model through adding in air recovery
- (c) Alternative nozzle position analysis
- (d) Alternative nozzle angle analysis
- (e) Alternative nozzle size analysis
- (f) Add front defrost as a parameter to future studies
- (g) Simulate worst-case wind scenario
- (h) Run simulation for varying external air temperatures

2. Future air curtain design considerations

- (a) Use third party vendor to supply air curtain
- (b) Create space for air curtain by modifying panels and or moving door mechanism
- (c) Determine method of mounting air curtain
- (d) Electrical and control system integration

3. Additional energy saving designs

- (a) Close-out door deadspace while door is in open position
- (b) Improve seal quality while door is in closed position
- (c) Damper on curbside front defrost during door cycling
- (d) Develop automatic width sensing doors

6.1 Future Air Curtain Analysis

Due to limited computing resources and abilities there were CFD studies that were planned out for the air curtain design, but were not completed. One study that is important to the proof of concept for the air curtain that should be conducted is the influence of passengers passing through the door. In its simplest form this would involve adding a block modelled person to the CAD model in the area of the open doors. A CFD study could then be conducted to see the effects of this disruption on the airflow being expelled from the air curtain. This study is being conducted with the goal of determining how the streamlines of the air curtain are impacted, Figure 1 gives a rough outline of the predicted area where the air curtains streamlines will be most affected.

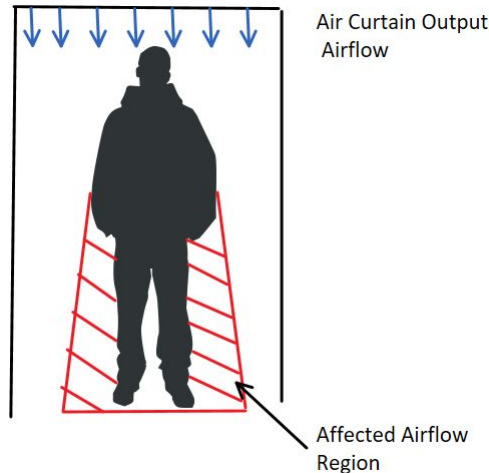


Figure 52: Predicted affected airflow region during passenger ingress and egress.

In the affected airflow regions, air will behave in a more turbulent fashion which will cause an increase in the amount of internally conditioned air that is able to escape. The increase in energy losses through this region will likely be partially balanced out by the fact that a passenger is a solid and there will be no chance for airflow in the space they are taking up. Passengers will constantly be interrupting the air curtains flow while in operation thus the steady state airflow will be frequently disrupted and need time to re-stabilize. Overall, the purpose of this CFD study would be to look at how the heat transfer rate through the door is altered during passenger ingress and egress as well as how this affects the heat transfer in the time after they pass through the plane of the air curtain.

Through discussion, it has been predicted that during passenger loading the effectiveness of the air curtain will be reduced which will ultimately reduce the efficiency of the air curtain. This loss is expected to only be minimal as some of the doors area will still have consistent airflow or be occupied by the solid that does not allow for air to escape through. If this prediction does not hold true and the effectiveness of the air curtain is severely reduced during passenger ingress and egress then alternative designs for addressing the projects main problem would need to be explored.

Note that this CFD study would be intensive to set up as the optimal study would require a moving mesh which is why it was unable to be implemented for the analysis within this project. An alternative to creating and conducting this complex study would be to conduct physical experimentation where the change in heat transfer through a doorway with an active air curtain is measured as a person passes through it.

Section 5.1.6 and Figure 44 detail one of the greatest issues with the current air curtain design that the air curtain streamlines are traveling outside of the bus with at an increasing rate as time goes on in the current model. It is ideal to pump interior cabin air through the plenum as it will relate directly to one of the key project target specifications in maintaining or improving passenger comfort. This is because in cold weather

pumping the cold outdoor air would send chills down passengers spines as they pass through the air curtain which is not comfortable.

This first solution for this problem is to improve the CFD model in future analysis. Currently there is a large pressure build up internally over time as the bus model is condensed to only a small area and the air from the air curtain is not circulated or recovered in any way. This process causes a pressure build up which pushes the air curtain streamlines out of the bus which will directly result in energy losses. This can be addressed in the future by adding additional constraints to the study to more adequately represent how the air deposited from the air curtain will be circulated and recovered within the bus. This alone will drastically improve the effectiveness of the air curtain as a greater amount of the treated cabin air will be retained in the bus.

An additional method of addressing the issue of the air being released from the plenum leaving the confinements of the cabin is to conduct a study on the effect of the nozzle position and angle. The effect of nozzle position could be analyzed through conducting three CFD studies where the position is changed each time. The first study would be done at the edge of the door frame nearest the exterior. Then one could be conducted where the nozzle is at a middle position and finally one where the nozzle is moved to the in-board edge of the open door. Moving any further into the cabin would not help the air curtain as once it is outside the tunnel created between the open doors air could escape through the sides. Based on the current state of the analysis it is predicted that the most inboard nozzle position will be optimal as this will increase the distance air from the air curtain needs to travel to exit the bus which will reduce the chances of it leaving the bus.

Similarly, adding an angle to the direction of the air exiting the nozzle is another way of addressing this issue. Adding an inboard component to the velocity profile for the air curtain airflow would combat the forces that are pushing streamlines from the air curtain out of the bus. Testing for the ideal angle could be done by adding an angle of 5° to the nozzle until an optimal angle is found. Note that these studies and changes are being done based on the current CFD results and if releasing the pressure build up in the cabin changes the results then angling and or moving the nozzle in the curbside direction of the bus may prove to be the better design change.

Another design consideration that could be adjusted is the size of the nozzle opening. The current opening size was determined based on research on existing air curtains in different applications. This may not be optimal for the bus door application and could be explored by altering the nozzle opening size in different CFD studies.

There are several conditions that exist in the environment that buses operate in that were not explored in the CFD that was completed. These changes should be explored as they could have an impact on the results. The first condition that should be added to the study is to add another parameter for the airflow from the front window defrost. The air from the front window defrost is heated as it needs to fulfill its job of maintaining visibility through the window. This air will circulate around the bus and since it is expelled near the door it will interact with the airflow passing in and around the entrance door. A schematic of where this air is released into the cabin is shown in Figure 53.

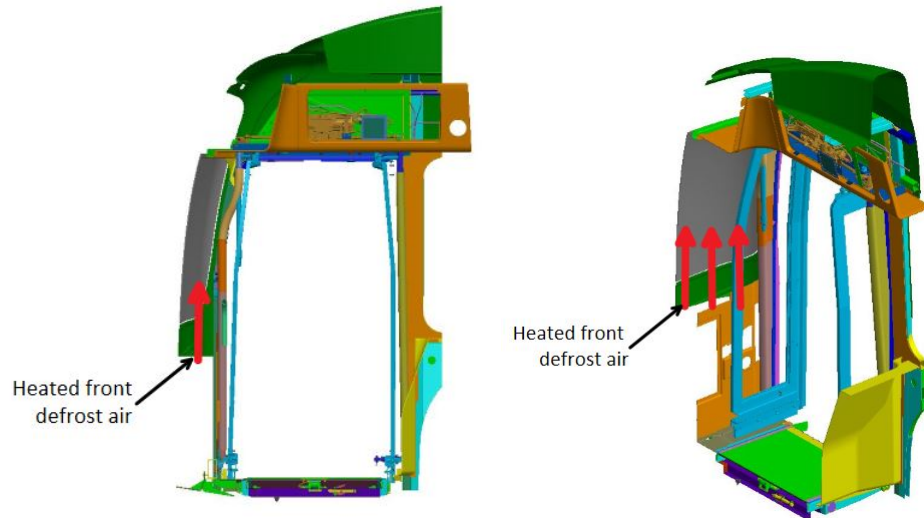


Figure 53: Location of curbside front defrost air.

Adding this output parameter to both the energy calculations and CFD model will impact the outlook on the effectiveness of the air curtain. Since in the current bus setup this air is likely sucked directly out of the bus during passenger loading the air curtain could have even more energy savings that originally calculated as this is heated air. Alternatively, adding this constraint to the study could show that more heat transfer is happening through the deadspace area than anticipated, refer to Figure 1.

Another factor to consider in future CFD analysis is a worst case wind scenario. Buses primarily operate in major cities where large buildings can create wind tunnels down streets. Thus, completing a study to analyze how a 60km/hr gust will interact with the air curtain would be valuable data to have. These winds would apply a force on the air curtain air stream that would angle the air into the bus. If the force applied by the wind pushes the air curtain stream lines too far inboard then the nozzle position and angle can be adjusted to account for this scenario. Furthermore, since the air speed of the air curtain is faster than it needs to be the wind should not be able to penetrate through it. This would need to be confirmed using numerical analysis or physical testing to simulate the scenario.

The studies conducted were done using -20°C as the set external air temperature as that is the lowest average temperature which occurs in Winnipeg in January. Further studies should be performed to see how the rate of heat loss through the door changes in extreme cold temperatures such as -40°C and less cold temperatures like -5°C . CFD analysis on hot weather climates where the external temperature would be set at 45°C . Air curtains are often used industrially to maintain the temperature of refrigeration units, hence the air curtain should remain efficient in these temperatures. Ultimately, running CFD tests for different temperatures will create a dataset that can be used when fine tuning the control system of the air curtain because this data will show the temperature range when activating the air curtain would have diminishing returns.

6.2 Future Air Curtain Design Considerations

Once the further analysis outlined above has been complete and the specifications required for the air curtain are finalized the next phases of the design must be considered. Given the complexity of the analysis detailed designs for the next phases of the design were unable to be completed in depth but were still given consideration. The major focus of the next part of the design process would be the integration of the air curtain into the current bus design space.

An excellent option for this project would be to purchase the air curtain from a third party vendor. This can serve as a less expensive option as NFI could purchase an existing or modified air curtain from a third

party vendor which would reduce the amount of time a money spent on making the entire assembly in house. Through discussion with the client this is a realistic avenue to explore as NFI outsources lots of parts and systems to third party vendors in order to reduce the amount of stress put on the NFI manufacturing facilities. For example, the air curtains produced by [4] could likely be modified to be the proper size and output velocity for the bus application. Financially vendors would be willing to make changes to their product as required as NFI would be purchasing a large quantity of these air curtains over time.

Depending on where the optimal final position is determined to be several components of the bus will need to be modified and or moved to accommodate for the space claim of the plenum and the air curtain intake duct. Figure 54 highlights the two main parts that would need to be modified to accommodate for the addition of an air curtain to the bus.

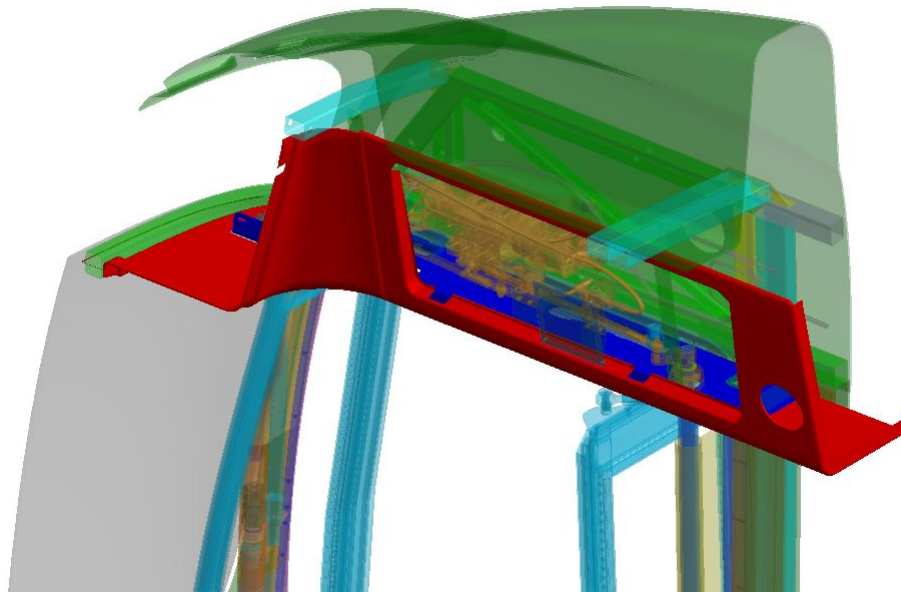


Figure 54: Entrance door top panel (blue) and top baseplate (red) isometric view.

Figure 55 shows a top view of the approximate area, highlighted in yellow, where a cut-out will be needed to allow for the nozzle of the air curtain to pass through the top panel, in red.

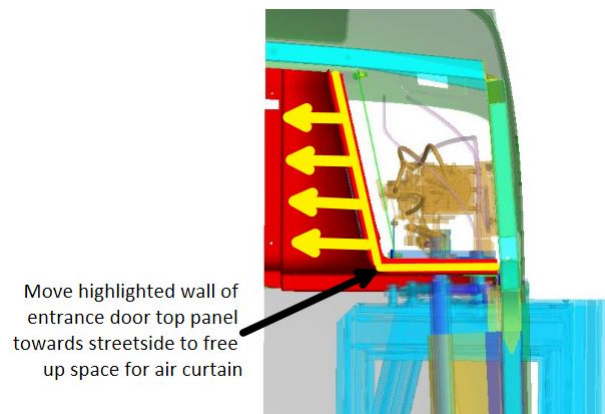


Figure 55: Entrance door top panel new shape.

Figure 56 shows a rear view of the entrance door mechanism and panels. In this figure the highlighted wall will need to be moved towards the streetside of the bus to open up space for the air curtain plenum to fit in.

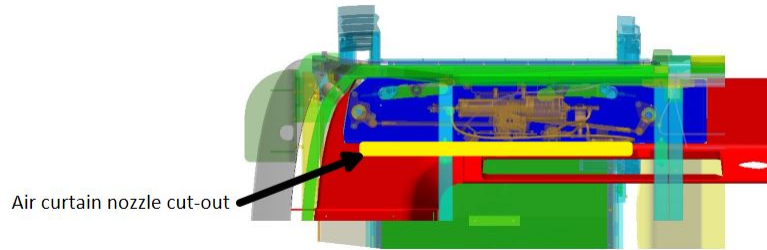


Figure 56: Cut-out region for plenum nozzle in entrance door top panel.

In addition to the changes outlined above the panel will require cutouts for the plenum and plenum duct. Further changes will need to be made to re-surface the top panel to keep the part smooth and aesthetically pleasing. Overall, the changes made to these parts would have downstream effects on the assemblies and installs that they are a part of.

Moving the door mechanism itself is another option that could be used to aid in creating more space for the air curtain. This could be accomplished by shifting the door mechanism up or further into the bus. Moving the mechanism would create a chain of additional engineering work that would need to be completed to make the change. Moving or lengthening of the mechanism arms would alter the performance of the door and further life cycle and finite element analysis would be required to ensure the integrity of the door mechanism is maintained.

To successfully mount the air curtain several brackets would need to be designed. Together these brackets would need to support the weight of the air curtain, hold the air curtain in place and attach to existing frame members of the bus. This design process would require structural analysis using a finite element analysis and an understanding of how to design for production.

Finally, the electrical and control system components would need to be integrated into the system. This process would require an analysis of the ideal operating range for the air curtain which is discussed in Section 5.3. Altogether these changes to integrate the air curtain would require a considerable amount of engineering time to ensure NFI product lifecycle management requirements are met.

6.3 Additional Energy Saving Designs

Aside from the air curtain there were a few additional design ideas generated during the project that also have potential to reduce energy losses through the door. Using the existing CFD model small changes could be made to analyze whether these alternative design concepts could prove to be effective ways of improving energy usage in the bus in ways that the air curtain could not.

One design idea that has already been discussed in this report is the deadspace close-out, see Section 1.6.1. To analyze this design idea a thin wall could be modelled in the deadspace region, defined in Figure 1, near the outer shell of the bus. Adding this blockage would seal off space that currently allows for heat transfer when the door is open. CFD analysis on this design idea should be done after the CFD model is more fully developed to include the front defrost HVAC outlet as the airflow from this outlet likely escapes through this deadspace region.

The design of this deadspace close-out itself can be executed in many ways. One simple way of doing this would be to spring load panels that are cut in the shape of the deadspace region. Whenever the door is closed the spring would be fully compressed by forces created by the door mechanism, then as the door opens this force is released and the spring stretches out pushing the panels into place to seal the air in that

region.

Another design idea that could be explored to make further improvements to the energy efficiency of the door is to conduct study on the seal of the doors when closed. As mentioned in Section 1.6.2 this improving the seals would require the use of lean six sigma principles: define, measure, analyze, improve, and control. Since the problem is already well defined the next step would be to measure it. The first step in measuring would be to determine how much heat is actually lost through gaps in the area of the door. Using Figure 57 it can be seen that a low pressure region exists near the entrance door of the bus.



Figure 57: Dynamic Pressure Analysis of a Bus [20].

The exact pressure measurement for the bus in this region could be obtained by conducting CFD on the busses external shell when it is moving or conducting physical testing using a pressure measuring device. With a proper understanding of the internal conditions, from CFD base case, and external conditions, pressure and temperature, analytical calculations could be conducted to measure the heat loss through a small gap between the two regions. Based on this measurement it could be determined whether or not a full study would be worth the investment. The next thing to measure would be how these issues with the seal of the door come to be. This could be done through looking a sample busses of various usage levels and trying to create a relationship between wear characteristics and time. The key areas to look at would be the rubber and brush seal degradation over time and the fitment of the door due to applied stresses over time. Then using this data a statistical analysis could be conducted to see where and when these issues start to cause major energy losses. The final stages of the lean six sigma process would be to address the finding above through altering part materials, size, and or the maintenance cycle of these parts. Improving these areas would help to eliminate the propagation of leaks when the door is closed, hence reducing the energy loss.

If the CFD study with the front defrost added into it proves to be a major source of heat loss and the deadspace closeout does not address this well then a design option would be to alter the current defrost system. This could be done through either re-routing the air deposited by this system or dampening it temporarily. Both of these concepts would be controlled similar to how the air curtain would be in that it actuates when the door cycle starts and stops when the door returns to a closed position. The re-route would have adjustable louvers that change their position to direct the air coming out of them away from the door. The dampening method would stop or reduce the flow of air coming from the defrost system. If the defrost systems function maintaining driver visibility is reduced then this design idea would be invalid. Thus, validation of this design concept would require physical testing to ensure that the front window can remain clear of frost even with the change to the defrost system.

7 Conclusion

In conclusion, the objective of the project was to design a door system that reduced energy loss during passenger ingress and egress through the bus door. Extensive numerical analysis was performed on both the current door system (base case) and the proposed air curtain solution.

The team was able to successfully set-up and quantify the amount of heat transfer occurring through the passenger bus door during a 7 s transient door cycle. The base case simulation found a heat transfer rate of 97 kW, compared to 380 kW with the addition of the air curtain. The observed reduction in performance was attributed to the lack of an HVAC return in the simplified model, and the air curtain air crossing the door boundary at an increased rate due to pressure build up within the bus cabin. Although the results show a worse heat transfer rate with the air curtain operating, the simulations used conditioned air in the air curtain. Therefore, the mass flow rate exiting the air curtain plenum that crossed the door boundary significantly increased the heat transfer rate observed in STAR-CCM+. Through the optimization of the air curtain nozzle parameters such as area, nozzle angle, and door clearance, it is expected that the measured heat transfer rate would reduce significantly. Our simulations successfully demonstrated the effectiveness of the air curtain in separating the conditioned internal HVAC air from the ambient air. This shows that with fine tuning, the air curtain is a viable solution with real world potential in commercial electric transit buses.

Further, justification that the design concept would work with more tuning and analysis stems from the fact that the fan power usage was equivalent to only 0.67 km of battery range during a 12 hour operating period. Even if the assumptions made for this calculation are inaccurate the result would still be far less than the 160.4 km of battery range lost in an average January day in Winnipeg.

Additionally, it was confirmed that it is possible to package a system in the existing bus envelope that would meet the required volumetric flow rates to achieve a proper boundary at the door orifice. Thus, virtually preventing the exchange of air at the door boundary.

Although the air curtain makes sense in practice, in reality it is a complex problem to model accurately. This project emphasizes the importance and sensitivity to the set-up of complex flow simulations. This is observed with the final design blowing a significant amount of air through the door boundary. While, the project fell slightly short of reducing the energy loss through the door during passenger ingress and egress, it remains a useful exploration into the potential of utilizing air curtains on commercial transit buses. Future recommendations as to how this concept could evolve into a design ready for implementation were presented. This includes a more in depth study on how various parameters affect the analysis, and determining the optimal combination of related design features. Overall, there was significant discovery surrounding the importance of set up parameters and design elements in order to produce a successful air curtain design.

8 References

- [1] New Flyer Industries. “About new flyer.” (), [Online]. Available: <https://www.newflyer.com/company-new/about-new-flyer/>. [Accessed: 2022-11-15].
- [2] *Standard Bus Procurement Guidelines*, American Public Transportation Association, 1300 I St. NW, Washington D.C., 2020, p. 354.
- [3] Siemens. “Star-ccm+ user guide.” (Jan. 2022), [Online]. Available: https://docs.sw.siemens.com/en-US/doc/226870983/PL20220315376513299.starccmp_userguide_html?audience=external. [Accessed: 2022-11-00 - 2022-12-07].
- [4] airtechnics. “Air curtain optima.” (), [Online]. Available: <https://www.airtechnics.com/products/air-curtain-optima>. [Accessed: 2022-12-04].
- [5] NISCO. “A closer glimpse at the working principle of axial fans.” (), [Online]. Available: <https://nisco.ca/a-closer-glimpse-at-the-working-principle-of-axial-fans/>. [Accessed 2022-11-25].
- [6] NISCO. “Clean air axial fans.” (), [Online]. Available: <https://nisco.ca/category/products/clean-air-axial-fans/>. [Accessed 2022-11-26].
- [7] AIR TECH SYSTEM. “Functionality of tangential fans / application solutions.” (), [Online]. Available: <https://www.ltg.de/us/products-and-services/ltg-process-air-technology/fans/tangential-fans/functionality-of-tangential-fans/>. [Accessed: 2022-12-04].
- [8] NISCO. “Differences between axial and centrifugal fans.” (), [Online]. Available: <https://nisco.ca/differences-between-axial-and-centrifugal-fans/>. [Accessed 2022-11-28].
- [9] C. Shu., L. L. Wang, C. Zhang, and D. Qi, “Air curtain effectiveness rating based on aerodynamics,” *Building and Environment*, vol. 169, p. 106582, 2020. DOI: <https://doi.org/10.1016/j.buildenv.2019.106582>.
- [10] Engineering ToolBox. “Air curtains or air screens.” (2003), [Online]. Available: https://www.engineeringtoolbox.com/air-curtains-d_129.html. [Accessed 2022-11-25].
- [11] Weather Spark. “January weather in winnipeg.” (), [Online]. Available: <https://weatherspark.com/m/8367/1/Average-Weather-in-January-in-Winnipeg-Canada>. [Accessed: 2022-11-15].
- [12] Weather Spark. “July weather in phoenix.” (), [Online]. Available: <https://weatherspark.com/m/2460/7/Average-Weather-in-July-in-Phoenix-Arizona-United-States>. [Accessed: 2022-11-15].
- [13] G. Chiriac, D. D. Lucache, C. Nițuță, A. Dragomir, and S. Ramakrishna, “Electric bus indoor heat balance in cold weather,” *Applied Science*, vol. 11, no. 24, pp. 95–109, 2021. DOI: <https://doi.org/10.3390/app112411761>.
- [14] City of Phoenix. “Will a lower speed limit help reduce speeding?” (), [Online]. Available: <https://www.phoenix.gov/streets/safety-topics/speed-limit#:~:text=...in%20Phoenix%20is%2055,15%20mph%20limit%20in%20alleys>. [Accessed: 2022-11-15].
- [15] Winnipeg Transit. “Blue rapid transit line.” (), [Online]. Available: <https://info.winnipegtransit.com/en/service/blue-rapid-transit/>. [Accessed: 2022-12-02].
- [16] F. Haghighat, J. Rao, and P. Fazio, “The influence of turbulent wind on air change rates—a modelling approach,” *Building and Environment*, vol. 26, no. 2, pp. 95–109, 1991. DOI: [https://doi.org/10.1016/0360-1323\(91\)90017-6](https://doi.org/10.1016/0360-1323(91)90017-6).
- [17] Leap Australia. “Turbulence part 2 - wall functions and y+ requirements.” (Jan. 2022), [Online]. Available: <https://www.computationalfluidynamics.com.au/tips-tricks-turbulence-wall-functions-and-y-requirements/>. [Accessed: 2022-12-04].
- [18] MOUSER ELECTRONICS. “Nmb technologies f280a3-113-d0520.” (), [Online]. Available: <https://www.mouser.ca/ProductDetail/MMB-Technologies/F280A3-113-D0520?qs=4W3PNmGdEWHN8MKsAr69Mw%3D%3D>. [Accessed: 2022-12-02].

-
- [19] airtecnics. “Air curtains control and regulation.” (), [Online]. Available: <https://www.airtecnics.com/technology/air-curtains-control-and-regulation>. [Accessed: 2022-12-04].
- [20] O. Arteaga, *Aerodynamic optimization of the body of a bus*. [Online]. Available: <https://iopscience.iop.org/article/10.1088/1757-899X/872/1/012002/pdf>.

Appendix A Centrifugal Fan Specifications

Specifications	
Product Attribute	Attribute Value
Manufacturer:	NMB Technologies
Product Category:	Blowers & Centrifugal Fans
RoHS:	 Details
Current Type:	DC
Operating Supply Voltage:	24 VDC
Airflow:	1166 CFM (33 m3/min)
Bearing Type:	Ball
Noise:	77 DBA
Speed:	2150 RPM
Power Rating:	175.2 W
Height:	280 mm
Depth:	280 mm
Width:	129 mm
Housing Material:	Plastic
Termination Style:	Wire Leads
Series:	F280
Brand:	NMB Technologies
Pressure Type:	1.91 in H2O
Product Type:	Motorized Impellers
Factory Pack Quantity:	4
Subcategory:	Fans, Blowers & Accessories
Tradename:	NMB-MAT
Unit Weight:	2.1 kg

Figure A.1: Specification for centrifugal fan.
[18]

Appendix B manufacturing Quote

B.1 Design manufacturing quote

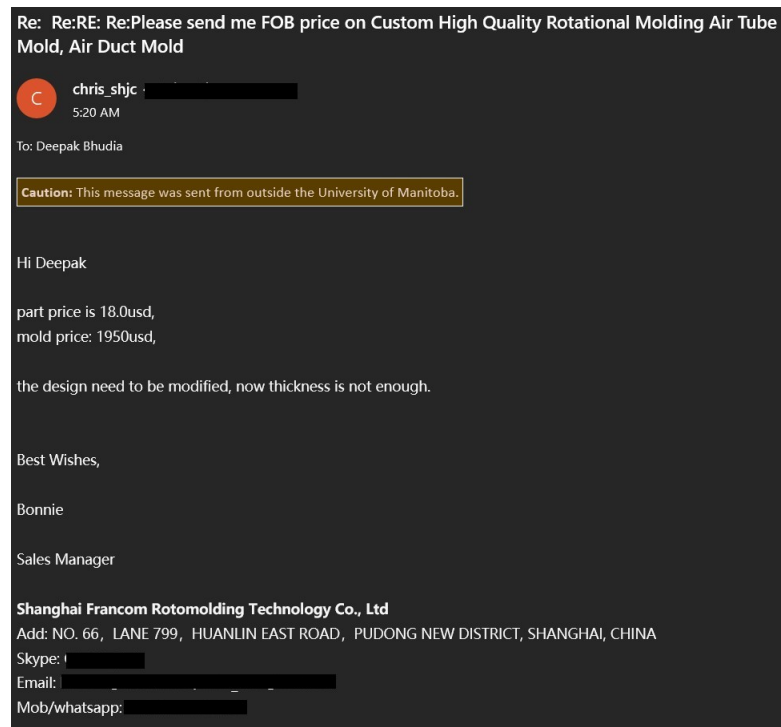


Figure B.1: Price quote from the manufacturing.

B.2 Roto-molding charges

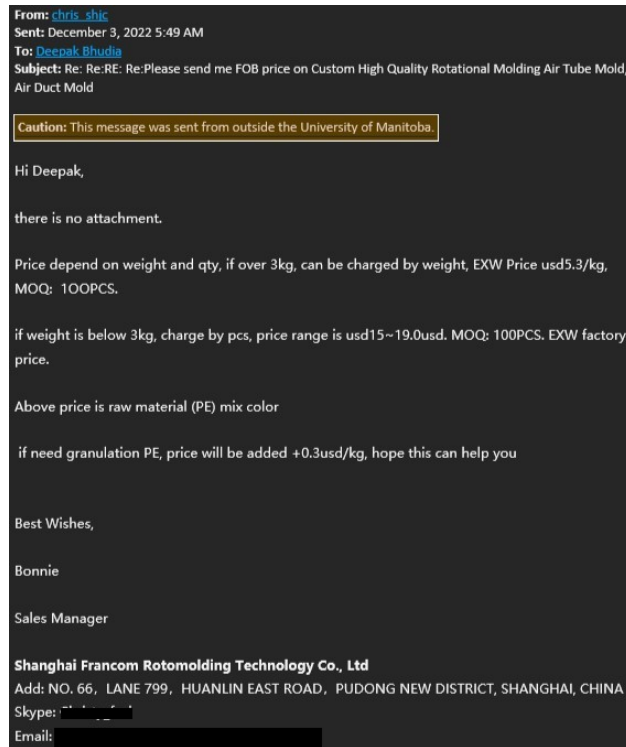


Figure B.2: Plum Roto-molding charges.

Appendix C FMEA Rubrics

C.1 Severity of Effect

Table XIX: SEVERITY OF EFFECT

Severity of Effect		Ranking
Minor	Unreasonable to expect that the minor nature of this failure would cause any substantial effect on system performance or on a subsequent process or service operation. Customer unlikely to either notice or care about the failure	1
Low	Low severity ranking due to nature of failure causing only a slight customer annoyance. Customer will notice only a minor degradation of the service performance, or a slight impact on a subsequent action, i.e., some quick, minor rework	2,3
Moderate	Failure causes some customer dissatisfaction. Customer is made uncomfortable or is annoyed by the failure. Customer will experience some very noticeable inconvenience or performance degradation. May cause either delay due to rework or irreversible damage	4,5,6
High	High degree of customer dissatisfaction due to negative impact of the failure such as an inaccurate payroll run, loss of vital data or an inoperable convenience system (i.e., computer crashes). Does not involve safety or noncompliance to government regulations. may cause serious disruption to subsequent processing; may require major rework or loss of customer and /or create significant financial hardship	7,8
Very High	Failure mode involves serious personal safety hazards, potential for civil litigation or noncompliance with government regulations	9,10

C.2 Failure Probability

Table XX: FAILURE PROBABILITY

Probability of Failure		Ranking	Possibility Failure Rates
Remote	Failure is unlikely. No failure ever associated with almost identical process	1	<1 in 20,000
Very Low	Process is in statistical control. Only isolated failure associated with almost identical processes	2	1 in 20,000
Low	Process is in statistical control. Isolated failures associated with similar processes	3	1 in 4000
Moderate	Generally associated with processes like previous processes which have experienced occasional failures, but not in major proportions. Process is in statistical control	4	1 in 1000
		5	1 in 400
		6	1 in 80
High	Generally associated with processes similar to previous processes that have often failed. Process is not in statistical control	7	1 in 40
		8	1 in 20
Very High	Failure is almost inevitable	9	1 in 8
		10	1 in 2

C.3 Likelihood of Risk Detection

Table XXI: LIKELIHOOD OF RISK DETECTION

Likelihood of Detection		Ranking
Very High	Current controls will almost certainly prevent the failure (process automatically prevents most failures)	1,2
High	Current controls have a good chance of detecting the failure	3,4
Moderate	Current controls may detect failure	5,6
Low	Current controls have a poor chance of detecting the failure	7,8
Very Low	Current controls probably will not detect the failure	9
Absolute certainty of non-Detection	Current controls will not or cannot detect the failure	10

Appendix D Engineering Drawing

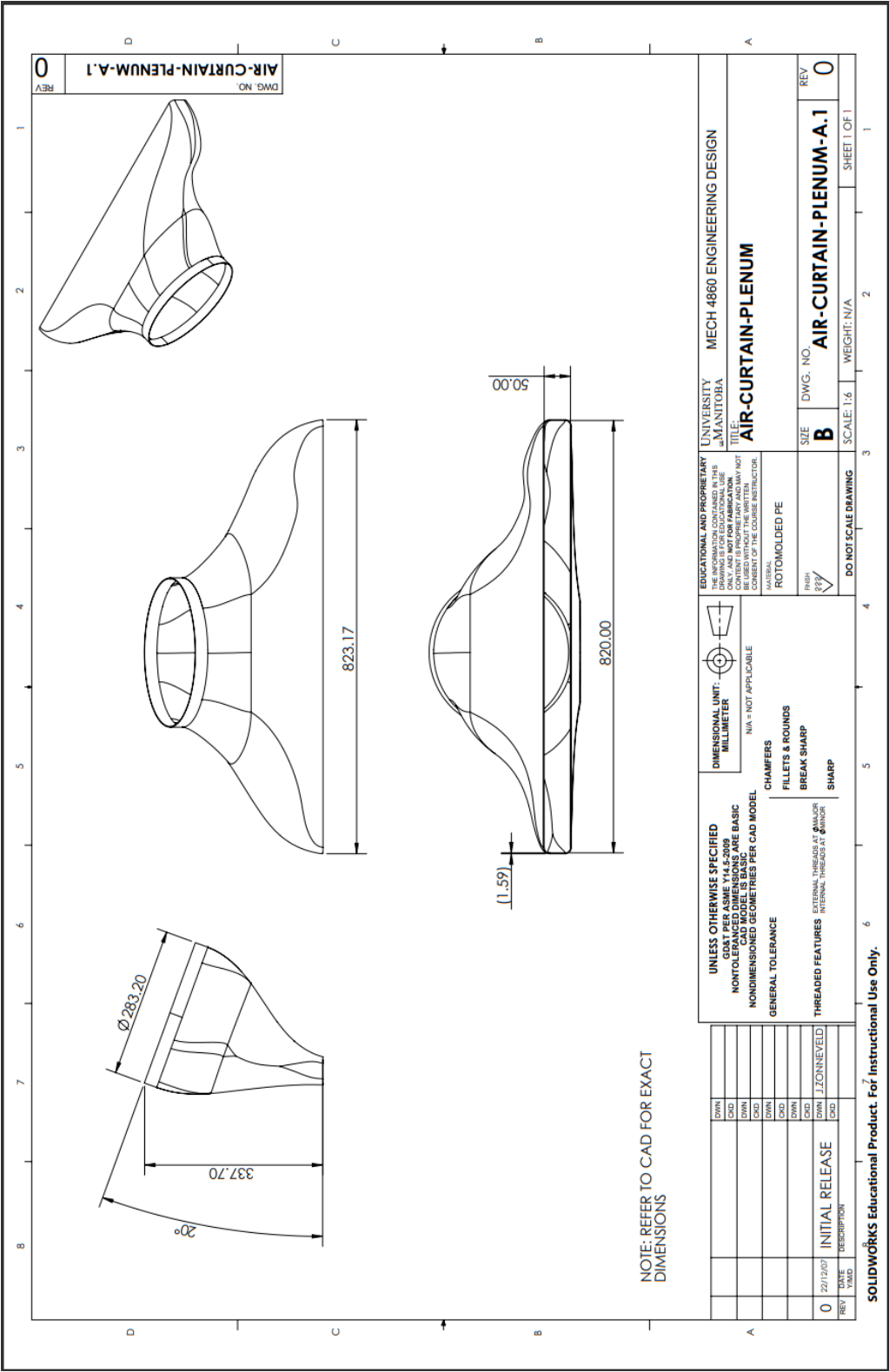


Figure D.1: Air curtain plenum drawing.