

To the Graduate Council:

I am submitting herewith a thesis written by Richard Jackson Holder entitled “The Effect of Bluff Body Flameholders on the Inter-Turbine Burner Performance.” I have examined the final electronic copy of this thesis for form and content and I recommend that it be accepted in partial fulfillment of the requirements for the degree of Master of Science with a major in Mechanical Engineering.

Dr. Cheng-Xian Lin, Major Professor

We have read this thesis
and recommend its acceptance:

Dr. Cheng-Xian Lin

Dr. Allen Baker

Dr. Kwai Wong

Accepted for the Council:

Carolyn R. Hodges,
Vice Provost and Dean of the Graduate
School

(Original signatures are on file with official student records.)

THE EFFECT OF BLUFF BODY FLAMEHOLDERS ON THE INTER-TURBINE BURNER PERFORMANCE

A Thesis
Presented for the
Masters of Science
Degree
University of Tennessee Knoxville

Richard Jackson Holder
August 2008

Abstract

In this thesis, a three-dimensional numerical simulation has been conducted to study the complex reactive flows which are present during the combustion in an inter-turbine burner (ITB) with the inclusion of V-gutter flame holders. Optimal conditions for the V-gutter were then further studied through a parametric study. The ITB configuration with straight radial vanes (SRV) was based on the innovative, high efficiency, high-g Ultra-Compact Combustor (UCC) concepts developed at the Air Force Research Laboratory (AFRL).

For the first section, the V-gutter's angle of attack was varied from -10 degrees to 10 degrees. The turbulent flow was modeled with a RANS-based realizable k-epsilon turbulence model, while the Jet-A spray combustion is modeled with the eddy-dissipation model. Numerical results indicate that the V-gutter not only generated vortices behind itself, but also altered the turbulent flow features and mixing behavior between main air flow and the circumferential and SRV cavity flows within the ITB. The exit temperature profile of the ITB could be modified substantially by the inclusion of the V-gutters at different angle of attack. The additional pressure drop incurred by the addition of the V-gutter was found to be less than 1 percent. Details of the vane cavity dynamics and increased entrainment physics were also discussed in the paper.

In the second section, a numerical parametric study has been conducted to examine the effects of varying both inlet turbulent intensity and angle of attack for a bluff body flame holder (V-gutter) in a cavity. The geometry used was based on previous experimental work. The inlet turbulent intensity was varied from 2 percent to 100 percent while the angle of attack of the V-gutter was varied from 0 degrees to 30 degrees. The combustion setup used was premixed propane-air combustion with an equivalence ratio of 0.6. The same numerical models which were used in the first section were also used here. Results indicated that increasing the inlet turbulent intensity and V-gutter angle of attack resulted in an increase not only in the size but also in magnitude of the downstream high turbulence areas with vortices.

Acknowledgements

There are many people without which this thesis would not have been possible. First and foremost, I would like to thank my advisor, Dr. Charlie Lin, for giving me the opportunity perform all of this research. Without him, I would not have had any exposure to the world of CFD.

I would also like to thank everyone involved with the facilities I used at Wright Patterson Air Force Base. There are countless people who make the computing resources there some of the best in the world. More specifically, I would like to thank Dr. Balu Sekar for providing me with much guidance on this project, the contracts for the work performed at WPAFB, and making sure to remind me to “get back to work.” In addition, many thanks go out to Dr. Hugh Thornburg, who provided many hours of on site troubleshooting for a wide variety a computing and mesh generation difficulties. Without him, this research would have taken much longer than it already did take. I would also like to thank Fernando Browning, Chris Chang, and Darius Sanders for keeping me company in the classroom and providing assistance and advice during post-processing. Also, thank you too University Technology Corporation for providing me with the funding and opportunity to return to WPAFB for a second summer.

Last, but certainly not least, I would like to thank my wife, Ginger Holder, and my parents, Jim and Sharyn Holder, for all their patience, understanding, and support during this research. Without them, this wouldn’t have been possible.

Richard J. Holder

Table of Content

I. Introduction	1
1.1 V-Gutter Inclusion in ITB Design.....	1
1.2 Parametric Analysis of V-Gutter	2
II. Background	4
2.1 Previous ITB Research	4
2.2 Previous V-Gutter Research.....	5
2.3 Thesis Objectives	7
III. Mathematical Models.....	8
3.1 Governing Equations	8
3.2 Turbulence Model.....	9
3.3 Combustion Model	10
3.4 Spray Injection Model	11
3.5 Thermophysical Properties	12
IV. Numerical Methods	13
4.1 Pressure-Velocity Coupling	13
4.2 Discretization	15
V. 3D ITB Study	16
5.1 Physical Model.....	16
5.2 Boundary Conditions.....	17
5.3 Convergence.....	18
5.4 Computational Mesh	19
5.5 Results and Discussion.....	20
VI. V-Gutter Parametric Study	26
6.1 Physical Model.....	26
6.2 Boundary Conditions.....	26
6.3 Convergence.....	27
6.4 Computational Mesh	27
6.5 Results and Discussion.....	28
VII. Conclusion.....	35
7.1 ITB Study.....	35
7.2 V-Gutter Parametric Study	36
7.3 Final Conclusions.....	36
References.....	39
Appendix I. ITB Figures and Tables.....	43
Appendix II. Parametric Study Figures and Tables.....	53
Appendix III. Comprehensive Baseline SRV ITB Results	64
Appendix IV. Curved Radial Vane (CRV) ITB.....	72
4.1 Introduction	72
4.2 Boundary Conditions.....	72
4.3 Convergence.....	73
4.4 Computational Mesh	73
4.5 Results and Discussion.....	74
4.6 Conclusion.....	77
Vita.....	78

List of Figures

Figure 1 Schematic of ITB Placement Within a Turbojet Engine.....	43
Figure 2 UCC Design Concept	43
Figure 3 Diagrams illustrating the air inlet locations and directions along with fuel injector positioning.	44
Figure 4 A 60 degree cross-section of the baseline configuration ITB.....	44
Figure 5 The 60-degree section ITB with the addition of the V-gutter.....	45
Figure 6 A close-up view of the V-gutter flameholder and the surrounding mesh.....	46
Figure 7 Detailed View of the V-Gutter with the SRV.....	46
Figure 8 Baseline ITB with labeled boundary conditions.	46
Figure 9 Axial slices of temperature (K) contours throughout the ITB domain.	47
Figure 10 Temperature (K) contours at the center plane inline with both the injector and the center of the V-gutter.	48
Figure 11 Temperature (K) contours at exit plane of ITB.....	48
Figure 12 Comparison of the mass averaged temperature profiles on the exit plane of ITB.....	49
Figure 13 Turbulent intensity contours on the periodic plane of the computational model.	49
Figure 14 Contours of vorticity on the periodic plane of the computational model.....	50
Figure 15 Comparison of relative pressure drop across the ITB.....	50
Figure 16 Profiles of Tangential Acceleration within the Circumferential Cavity of the SRV Baseline.....	51
Figure 17 Circumferential Interface Schematic.....	51
Figure 18 Mass Flow Rate Component Comparison	52
Figure 19 Original Geometry of V-gutter in a cavity.....	53
Figure 20 Validation Results for Turbulent Kinetic Energy extending from the V-gutter Centerline and the V-Gutter Y-max Axially Towards the Outlet [6]	53
Figure 21 Mixing Properties of Baseline Case Setup	54
Figure 22 Turbulent Intensity Contours at $\alpha = 0^\circ$ for Four Separate Inlet Turbulent Intensity Conditions (25%, 50%, 75%, and 100%).....	54
Figure 23 Turbulent Intensity Contours For Two Angles of Attack ($\alpha = 10^\circ$ and $\alpha = 20^\circ$).....	55
Figure 24 Turbulent Intensity Contours with $\alpha = 20^\circ$ for Four Separate Inlet Turbulent Intensity Conditions (25%, 50%, 75%, and 100%).....	55
Figure 25 Velocity Vector Plots Colored by Axial Velocity for Four Separate Inlet Turbulent Intensity Conditions (25%, 50%, 75%, and 100%)	56
Figure 26 Velocity Vector Plots Colored by Axial Velocity for Two Angles of Attack ($\alpha = 10^\circ$ and $\alpha = 20^\circ$)...	57
Figure 27 Velocity Vectors Colored by Axial Velocity with $\alpha = 20^\circ$ for Four Separate Inlet Turbulent Intensity Conditions (25%, 50%, 75%, and 100%).....	57

Figure 28 Temperature Contours of Baseline Setup Along with the Most Extreme Case of the Other Three Studies (Turbulent Intensity = 2%, $\alpha = 0^\circ$, Turbulent Intensity = 100%, $\alpha = 0^\circ$, Turbulent Intensity = 2%, $\alpha = 20^\circ$, Turbulent Intensity = 100%, $\alpha = 20^\circ$).....	58
Figure 29 CO2 Contours of Baseline Setup Along with the Most Extreme Case of the Other Three Studies..	59
Figure 30 Maximum Values of Temperature, Axial Velocity, and Turbulent Intensity	60
Figure 31 Friction Factor Calculated for Each Case with Lines of Constant Turbulent Intensity Shown.....	61
Figure 32 Wall Temperature Profile Comparison of the Upper and Lower Cavity Walls at $\alpha = 0^\circ$ and $\alpha = 30^\circ$	62
Figure 33 Wall Temperature Profile Comparison of the Upper and Lower V-Gutter Walls at $\alpha = 0^\circ$ and $\alpha = 30^\circ$	63
Figure 34 Wall Temperature Profile Comparison of the Rear V-Gutter Wall at $\alpha = 0^\circ$ and $\alpha = 30^\circ$	63
Figure 35 Contours of Propane Concentration for the Baseline Configuration.....	64
Figure 36 Contours of H2O for the Baseline Configuration.....	65
Figure 37 Contours of Turbulent Intensity for the Baseline Configuration.....	66
Figure 38 Contours of Rate of Reaction for the Baseline Configuration.....	67
Figure 39 Contours of Temperature for the Baseline Configuration	68
Figure 40 Contours of Axial Velocity for the Baseline Configuration.....	69
Figure 41 Contours of Radial Velocity for the Baseline Configuration.....	70
Figure 42 Contours of Tangential Velocity for the Baseline Configuration.....	71
Figure 43 Computational Mesh for Curved Vane ITB with V-Gutter.....	75
Figure 44 Mass Averaged Temperature Profile for the Curved Vane ITB With and Without V-Gutters.....	76
Figure 45 Temperature Contours of CRV ITB With and Without V-Gutters.....	76

List of Tables

Table 1 Baseline ITB Boundary Flow Conditions.....	47
Table 2 Mesh Convergence Study	53
Table 3 Max Value Comparison of Baseline with the Most Extreme Case of the Other Three Studies	59
Table 4 Recirculation Zone Size Differences for Varied Inlet Angle of Attack and Turbulent Intensity.....	60

List of Symbols

A_c	Flow cross-sectional area of test channel (m ²)
$\vec{A}$	Surface area vector
$\vec{A}_f$	Area of face f
a_p	Discretized coefficient for cell of interest
a_{nb}	Discretized coefficient for cells which neighbor cell of interest
a_t	Tangential acceleration (m/s ²)
B	Height of the V-gutter (=25 mm)
b	Source term of discretized continuity equation
c	Thermal conductivity (W/m·K)
C	Constants
D_h	Hydraulic diameter (m)
d_f	Average of the momentum equation coefficients for the cells on either side of face f
E	Total energy (J)
f	Friction factor
F	External body force term (N/m ³)
g	Gravity (m/s ²)
G_b	Generation of turbulence kinetic energy due to buoyancy (kg/m·s ³)
G_k	Generation of turbulence kinetic energy due to mean velocity gradients (kg/m·s ³)
I	Unit Tensor
J_f	Face flux
J_f^*	Face flux calculated from guessed pressure field
J_f'	Face flux correction term
$\hat{J}_f$	Contains the influence of velocities in the cells of interest
k	Turbulence kinetic energy (m ² /s ²)
L_{Rz}	Axial length of recirculation zone in experimental data (= 93.75 mm)
N_{faces}	Number of faces enclosing cell
p	Static Pressure (Pa)
p_{c0}	Pressure within adjacent cell of face f (Pa)
p_{c1}	Pressure within adjacent cell of face f (Pa)
p_{c0}'	Cell pressure correction
p_{c1}'	Cell pressure correction
R_i	Net rate of production by chemical reaction of species (kg/s·m ³)
S	Source term
S_i	Rate of creation by addition from the dispersed phase

	(kg/s·m ³)
t	Time (s)
T	Temperature (K)
V	Volume (m ³)
v	Velocity (m/s)
v_t	Tangential velocity (m/s)
x	Distance (m)
Y_i	Local mass fraction of chemical species
Y_M	Contribution of the fluctuating dilation in compressible turbulence to the overall dissipation rate (kg/m·s ³)

Greek Letters

α_p	Under-Relaxation factor
Γ_ϕ	Diffusion coefficient
∇	Gradient
ε	Turbulence kinetic energy dissipation rate (m ² /s ³)
ϕ	Property of interest
ϕ_f	Value of ϕ convected through face f
μ	Molecular viscosity (kg/m·s)
ρ	Density (kg/m ³)
$\bar{\tau}$	Stress tensor (Pa)
$\vec{v}$	Velocity vector ($u\hat{i} + v\hat{j} + w\hat{k}$)

Subscripts

b	Denotes Buoyancy
$c0$	Cell adjacent to face of interest
$c1$	Cell adjacent to face of interest
f	Face of interest
i	Species
M	Denotes fluctuating dilation in compressible turbulence
nb	Cells neighboring current cell
P	Current cell
Rz	Recirculation zone
t	Tangential

I. Introduction

1.1 V-Gutter Inclusion in ITB Design

Concepts such as reheat, regeneration, and intercooling have been developed for decades to increase the efficiency of the Brayton Cycle for gas turbine engines [1]. Previous studies have shown that secondary burning within the turbine engine can produce a much more favorable trade-off between specific thrust (ST) and thrust specific fuel consumption (TSFC) [2-5]. A schematic of this can be seen in Fig. 1. (All figures can be seen in Appendix 1.) However, implementation of a reheat burner is very challenging in the aerospace industries, because a jet engine has much stricter weight requirements as compared to a land-based gas turbine engine. The increased size and weight incurred from additional engine components could more than offset the benefits of increased engine efficiency. In order for the size and weight to only marginally increase, the length of the burner must be much shorter than conventional burners. Shortening the burner, however, decreases the residence time of the flow within the burner. This makes combusting all of the fuel within the burner much more difficult. Unburned fuel leaving the burner will severely damage the downstream turbine blades, which must be avoided [6]. The problems regarding the size, weight, and residence time were addressed by the new ITB concept being developed at the Air Force Research Lab (AFRL), called the ultra-compact combustor (UCC), which is shown in Fig. 2. This new combustion technology employed spray combustion in a circumferential cavity where air is injected at certain angle to generate high-g load flows [7-10]. The combustion products were further mixed with axial flow via the radial vane cavities

(RVC). Because the combustion occurred in a highly swirling flow assisted with enhanced mixing, the flame stability and residence time was increased within the cavity.

To address the above problems, the possibility of using a bluff-body V-gutter flameholder to improve the ITB performance was explored. This geometric configuration has never been studied before. Traditionally, flameholders are used in afterburners to aid in flame stabilization and shortening the required length of the combustion chamber [2]. Inclusion of V-gutter flameholders in an ITB has not been studied in the past. The objective was to determine whether the flameholder generated recirculation could increase the mixing between the high temperature circumferential flow and the low temperature main flow, resulting in a more uniform exit temperature profile as well as improved entrainment of combustion products to the main flow. Unlike the traditional V-gutter setup, there was no combustion taking place directly upstream of the V-gutter in the main flow. A large majority of the combustion takes place in the circumferential cavity. The study was performed by three-dimensional CFD simulation of the turbulent reactive flow in the ITB with V-gutter flameholder at several angles of attack. Numerical results of ITB with and without the V-gutters were compared.

1.2 Parametric Analysis of V-Gutter

Bluff body flame holders have been utilized in augmentor applications for many years. They have been proven to be very effective in anchoring the flame created in combustion environments [2]. With the ITB, there is now a less traditional application for the V-gutter within a combustor where mixing the flow is equally if not more important than flame holding [11].

It is well known that turbulent eddies create a very strong mixing effect [12]. Therefore, a V-gutter can also be used as an effective mixing device because it generates large amounts of

downstream turbulence. In order to optimize this for various applications, research was needed which determined the affect that varying different parameters had on the downstream turbulence.

In this section, the flame holder's mixing effects at different turbulence intensity and angle of attack were studied systematically. As the goal of this research was to study large scale results and overall trends, lower level turbulence (realizable $k-\epsilon$) and combustion (eddy-dissipation) models were employed to allow for shorter compute times. The parametric study was carried out to examine the changes in the mixture properties of a V-gutter due to changes in both angle of attack and turbulent intensity. The results were also analyzed to determine how these parameters affect the turbulent flow, size and location of the recirculation zones downstream of the V-gutter, and wall temperature within the domain.

II. Background

2.1 Previous ITB Research

During the last few years, numerical simulations have been carried out to investigate the UCC/ITB performance. One of the efforts was to address the issues related to the mixing of the circumferential and axial flows.

One of these works included implementing three different ramp configurations into the circumferential cavity to increase vorticity [13]. The idea was that increased vorticity within the circumferential cavity would increase the burning efficiency within the cavity and improve the exit temperature profile. It was found that efficient burning was achieved at all configurations of the ramps which indicated good flame stability and combustion efficiency. Also, different burning patterns were noticed downstream of the circumferential cavity, but these did not result in much improvement in the exit temperature profile.

Another study which was performed to optimize the ITB design involved testing the effects of three different RVC shapes on performance [14]. These three shapes included an angled rectangular cavity, forward facing, and backward facing step cavities. It was found that the RVC shape had a major impact on the ITB performance. A much smaller amount of combustion products were transported from the circumferential cavity into the main flow when the backward facing step cavity was used compared to the other two. These results were used to design the RVC which was used in this thesis.

Different radial vane shapes were also considered. A curved radial vane (CRV) replaced the SRV, and the two were compared [15]. This study found much improvement in the mixing of the combustion products with the main flow, resulting in more favorable temperature

gradients at the exit. However, these curved vanes also resulted in a higher pressure drop when compared to the SRV.

Further studies were carried out on the curved vane ITB which involved varying the equivalence ratio [16]. Equivalence ratio values of 0.12 and 0.44 were compared. It was found that the amount of combusted products was increased as the equivalence ratio was increased. This was due to the higher mass flow of fuel from the injectors which resulted in greater penetration into the main flow. The increased equivalence ratio also increased the pressure drop and decreased the g-loading within the cavity, though. Also, raising the equivalence ratio caused degradation in the quality of the exit temperature profile.

While the UCC/ITB addressed many of the problems which arose from the reheat concept, improvement in its current (baseline) design and performance was still needed. One of the major issues the basic ITB presented is the almost linear exit temperature profile with the maximum at the outer wall [15]. It is known that unburned fuel can negatively affect the downstream turbine blades [6], but an adverse temperature profile can damage the turbine blades as well. Therefore, an ideal temperature profile for the ITB exit is expected to be either uniform or parabolic [2].

2.2 Previous V-Gutter Research

Analysis of these devices has proven to be very difficult due to the highly turbulent nature of the flow downstream of the flame holder. Many experimental results have been found, but a large scale parametric analysis can be both expensive and time consuming.

One of these experimental papers included cold flow analysis of various flow characteristics [17]. The aerodynamic characteristics of a bluff body flameholder were obtained

experimentally. The recirculation zone and mixing of tracer gas with the surrounding airflow was observed. Also, a zone of opposing shear was formed near the region of co-existence between negative and positive mean axial-velocities.

Experimental results were also found for the case of a bluff body flameholder placed in reacting flow [18]. Analysis was performed on a bluff body flameholder using a laser Doppler velocimeter. It was found that the presence of combustion led only to quantitative changes of the mean structure of the reactive wakes when compared with their inert counterparts. However, a complete restructuring of the instantaneous flow structure induced by dramatic changes of the large-scale motion properties was observed when combustion was developed in the wake. This was compared to cold flow results.

A comparison of cold and reacting flows involving a V-gutter with a lean blow out analysis was also performed [19]. Large differences were found in the comparison between the two. Some of these included a suppression of turbulence observed in the separated combustion flow compared to the cold flow and a distortion of the turbulence structure at the visible flame location. This case was used to validate the numerical results.

CFD results have now become a reality with the increases achieved in computing power. Of these simulations, many have dealt more with validation studies than parametric analysis. One such result studied the accuracy of large eddy simulation (LES) modeling of a v-gutter in a cavity with reacting flow [20]. The LES model was found to have high accuracy, but the recirculation zone was slightly under predicted. The model was then used to perform a lean blowout analysis.

Another study performed LES simulation of a v-gutter in a cavity for both reacting and non-reacting flows [21]. These simulations were compared with experimental results, and once

again found that LES models can accurately predict the unsteady characteristics of the flow patterns created by the v-gutter. A comparison was also made with the RANS $k-\epsilon$ model was performed on the same geometry, and it was shown to have some decreased accuracy, which is to be expected.

An LES simulation was performed for both reacting and non-reacting flows and compared with experimental data [22]. The reacting case was modeled with two different closures for the filtered scalar transport equations. This study found that the eddy breakup model (EBU) creates a thicker flame and fails to adapt the turbulent fluctuations in the flow field. This caused an under-prediction of the turbulent flame wrinkling and the far-field wake spread.

Through the literary review, it was found that much research has been done involving the bluff body V-gutter's affect on combustion and turbulence, but none of this research has been focused on the turbulent mixing behaviors of these structures at different turbulence intensity and angle of attack.

2.3 Thesis Objectives

The overall goal of this thesis is to show the effect of the addition of a bluff body flameholder on the turbulence and combustion characteristics of the ITB. This will be carried out by first placing the v-gutter in the SRV ITB geometry and examining the results for a few different configurations. Then a full parametric analysis where both angle of attack and inlet turbulent intensity are varied will be performed. This will be used to help determine the optimal operating conditions for the v-gutter. The results of these two sections will then be examined to predict the benefits and deficiencies which arise from the placement of a v-gutter within the ITB geometry.

III. Mathematical Models

3.1 Governing Equations

The basic governing equations are the mass, momentum, and energy conservation equations. Conservation of mass is described by [23]:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \quad (1)$$

where the source term S_m includes any mass added to the continuous phase from the dispersed second phase (e.g., vaporization of liquid droplets). Conservation of momentum is as follows:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\bar{\tau}) + \rho \vec{g} + \vec{F} \quad (2)$$

where $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. The stress tensor $\bar{\tau}$ is given by:

$$\bar{\tau} = \mu \left[(\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \quad (3)$$

where the second term on the right hand side is the effect of volume dilation [23]. Conservation of energy is of the following form:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i} [u_i (\rho E + p)] = \\ \frac{\partial}{\partial x_j} \left[\left(c + \frac{c_p \mu_t}{Pr_t} \right) \frac{\partial T}{\partial x_j} + \mu_t (\tau_{ij})_{eff} \right] + S_h \end{aligned} \quad (4)$$

where c is the thermal conductivity, E is the total energy, and $(\tau_{ij})_{eff}$ is the deviatoric stress tensor, which is given by [23]:

$$(\tau_{ij})_{eff} = \mu_{eff} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \mu_{eff} \frac{\partial u_i}{\partial x_i} \delta_{ij} \quad (5)$$

3.2 Turbulence Model

The turbulence model chosen for this study was the Realizable k - ε model. The transport equations for turbulence kinetic energy (k) and its dissipation rate (ε) are:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \\ + G_k + G_b - \rho \varepsilon - Y_M + S_k \end{aligned} \quad (6)$$

and

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \\ + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1c} \frac{\varepsilon}{k} C_{3c} G_b + S_\varepsilon \end{aligned} \quad (7)$$

where

$$C_1 = \max \left[0.43 \frac{\eta}{\eta + 5} \right], \quad \eta = S \frac{k}{\varepsilon}, \quad S = \sqrt{2 S_{ij} S_{ij}} \quad (8)$$

The main difference between the Realizable k - ε and the standard k - ε model is the calculation of μ_t .

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (9)$$

The C_μ term is variable in the Realizable k - ε model, whereas it is a constant in the standard model [21]. It is defined by

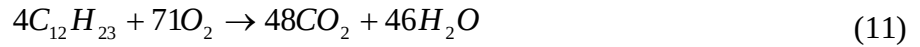
$$C_\mu = \frac{1}{A_0 + A_S \frac{k U^*}{\varepsilon}} \quad (10)$$

where $A_0 = 4.04$ and $A_S = \sqrt{6} \cos(\varphi)$.

Further studies will include LES simulation of unsteady effects. The motivation of this preliminary study was to better determine the ideal internal geometry using large scale effects. Therefore, a less robust, steady state turbulence model was chosen.

3.3 Combustion Model

In the first section, the Jet-A fuel combustion is simulated using a one-step irreversible chemical reaction:



While a single step reaction model may not be the most accurate, it is the least expensive computationally [23]. The objective of this paper was not to evaluate the most accurate combustion model. Since the main focus is on the flow mixing, the exactness of the reaction kinetics required did not warrant a more computationally intensive combustion model.

The second section was modeled the same as the first but used a different fuel. In this case, propane was used, which resulted in the following one-step irreversible chemical reaction:



The species transport which occurs as a result of this reaction is modeled using the following equation:

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla \cdot (\rho \vec{v} Y_i) = -\nabla \cdot \vec{J}_i + R_i + S_i \quad (13)$$

where Y_i is the local mass fraction of the species being calculated, S_i is the rate of creation by addition from the dispersed phase along with any user-defined sources, and R_i is the net rate of production by chemical reaction of the species being calculated.

The eddy dissipation model was used for the turbulence-reaction interaction. The net rate of production (R_i) value is calculated by taking the smaller value of these two equations:

$$R_{i,r} = \nu'_{i,r} M_{w,i} A \rho \frac{\varepsilon}{k} \min \left(\frac{Y_R}{\nu'_{R,r} M_{w,R}} \right) \quad (14)$$

$$R_{i,r} = \nu'_{i,r} M_{w,i} A B \rho \frac{\varepsilon}{k} \frac{\sum_p Y_p}{\sum_j^N \nu'_{j,p} M_{w,j}} \quad (15)$$

where Y_p is the mass fraction of the product being calculated, Y_R is the mass fraction of the reactant being calculated, A is an empirical constant ($A = 4.0$), and B is an empirical constant ($B = 0.5$) [23]. Since the second study used premixed combustion, only the species transport model was used to calculate the chemical interactions. The first study modeled spray combustion, though, so this must also be taken into account.

3.4 Spray Injection Model

In the 3 dimensional ITB study, spray injection was modeled to simulate real life conditions. The fuel injected into the ITB was modeled using a discrete phase model. This allows a simulation of a second discrete phase in a Lagrangian reference frame in addition to solving the transport equations in the continuous phase. The second phase consists of spherical particles released into the continuous phase. The particle trajectory is determined by solving the force balance:

$$\frac{du_p}{dt} = F_D(u - u_p) + \frac{g_x(\rho_p - \rho)}{\rho_p} + F_x \quad (16)$$

To allow turbulent effects to be factored into the motion of the discrete particles, the Discrete Random Walk Model was used. This model subjects the discrete particles to turbulent eddies

characterized by a Gaussian distributed random velocity fluctuation and a time scale. These particles were distributed using the Rosin-Rammler distribution. This divides the complete range of sizes into discrete intervals. Each interval is then given a mean diameter which was used to calculate the trajectory [23]. The discrete phase was set to interact with the continuous phase every 20 iterations. The spray conditions were set to model a real fuel spray closely. The fuel was sprayed with a velocity of 23.18 m/s in a cone shape with an angle of 35°. The injector was given a radius of 0.0014 m. This resulted in a mass flow rate of 0.000108126 kg/s.

3.5 Thermophysical Properties

When performing CFD calculations, the method in which thermophysical properties are calculated has a strong impact on the accuracy of results. In these simulations, methods which gave high orders of accuracy were used. When dealing with the fuel air mixture, the specific heat was modeled using the mixing law along with thermal conductivity and viscosity.

$$a = \frac{1}{\sum_i \frac{Y_i}{a_i}} \quad (17)$$

where a is the property being calculated. For each individual species modeled (air, O₂, CO₂, etc...), the specific heat and thermal conductivity was modeled with a piecewise polynomial approximation in terms of temperature, and the viscosity was modeled with the Sutherland approximation.

$$\mu = \frac{C_1 T^{\frac{3}{2}}}{T + C_2} \quad (18)$$

where C_1 and C_2 are material specific properties.

IV. Numerical Methods

4.1 Pressure-Velocity Coupling

The 3D ITB study used a pressure-velocity coupling scheme to solve the Navier-Stokes equations. This is achieved by deriving an equation for pressure from the discrete continuity equation. In order to do this, the equations were iterated in time, even for steady state solutions. This produces the following linearized system [23]:

$$\left[D + \sum_j^{N_{faces}} S_{j,k} \right] \Delta Q^{n+1} = -R^n \quad (19)$$

where

$$D = \frac{V}{\Delta t} \Gamma + \sum_j^{N_{faces}} S_{j,i} \quad \text{and} \quad S_{j,k} = \left(\frac{\partial F_j}{\partial Q_k} - \frac{\partial G_j}{\partial Q_k} \right) \quad (20)$$

For these equations, Q represents the variable being calculated, F represents a vector of forces, G represents a vector of stress tensors, Γ represents the preconditioning matrix, and R^n represents the residual vector. This method allows the user to specify a Courant number to control the time step used in the calculations.

$$C = \frac{\Delta t}{\Delta x_{cell} / v_{fluid}} \quad (21)$$

Doing so gives more control over the convergence of the solution. A larger Courant number results in a larger time step used for the iterations. If divergence occurs during the solution, one problem may be a Courant number which was too large. Therefore, altering the Courant number can help achieve convergence, but lowering this also causes the solution to converge more slowly.

The pressure-velocity coupling scheme, SIMPLE, was used for the second case. The SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. The following equation performs this task [23]:

$$J_f = \hat{J}_f + d_f(p_{c0} - p_{c1}) \quad (22)$$

If the momentum equation is solved with a guessed pressure field p^* , the resulting face flux computed from equation 16 does not satisfy the continuity equation. Therefore, a correction is added to the face flux so that the corrected face flux satisfies the continuity equation. The face flux is corrected by

$$J_f = J_f^* + J_f' \quad (23)$$

The SIMPLE algorithm states that the corrected face flux can be written as

$$J_f' = d_f(p_{c0}' - p_{c1}') \quad (24)$$

The algorithm then substitutes the flux correction equations into the discrete continuity equation to obtain a discrete equation for the pressure correction in the cell.

$$a_p p' = \sum_{nb} a_{nb} p_{nb}' + b \quad (25)$$

Once this equation is solved, the cell pressure and the face flux are corrected using

$$p = p^* + \alpha_p p' \quad (26)$$

$$J_f = J_f^* + d_f(p_{c0}' - p_{c1}') \quad (27)$$

As a result of the corrections, the face flux satisfies the continuity equation at each iteration [23].

4.2 Discretization

For this study, a finite volume method was used to convert the governing equations to algebraic equations that could be solved numerically. This control volume technique consisted of integrating the governing equations about each control volume. This resulted in equations that conserved each quantity on a control-volume basis. In doing this, equations of the following form were integrated across each individual control volume [23].

$$\oint \rho \phi \vec{v} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} + \int_V S_\phi dV \quad (28)$$

Once this equation is discretized on every cell in the domain, the following equation is developed:

$$\sum_f^{N_{faces}} \rho_f \vec{v}_f \phi_f \cdot \vec{A}_f = \sum_f^{N_{faces}} \Gamma_\phi (\nabla \phi)_n \cdot \vec{A}_f + S_\phi V \quad (29)$$

By default, the discrete values of ϕ at the cell centers. However, face values are also required for the convection terms in the previous equation. This requires interpolation. For the cases discussed, this is accomplished using the second order upwind scheme. In the second order scheme, the face value ϕ_f was computed using the following expression [23]:

$$\phi_f = \phi + \nabla \phi \cdot \Delta \vec{s} \quad (30)$$

where ϕ and $\nabla \phi$ are the cell-centered value and its gradient in the upstream cell, and $\Delta \vec{s}$ is the displacement vector from the upstream cell centroid to the face centroid.

V. 3D ITB Study

5.1 Physical Model

Fig. 3 shows the overall arrangement of the fuel and air flows in the ITB, with orientation of all the fuel and air jets around the circumferential cavity depicted in details. Due to the angled injection of air from the outer wall of the circumferential cavity, the centrifugal forces can generate high-g load flow. JET-A fuel is injected into the cavity in the radial direction, and spray combustion is initiated due to hot air in the cavity, resulting in swirling flames. As the geometry is either symmetric or periodic, only 60-degree section of the ITB was used in the actual computation.

The full axial length of the V-gutter geometry is 0.149 m with a diameter of 0.0768 m upstream of the circumferential cavity and 0.1176 m at the cavity. This results in a circumferential cavity that is 0.0204 m tall. The circumferential cavity is also 0.047625 m long in the axial direction. The radial blade which runs axially down the combustor measures 0.11716 m.

A 3-D representation of the computational domain with unstructured surface meshes used in this study is shown in Fig. 4. Within each 60-degree section, two pairs of inlets inject air at 45 and 30 degree angles to the normal direction, respectively. Inside the ITB there are radial vanes which extrude radially from the hub of the turbine and meet the bottom boundary of the circumferential cavity. For the baseline configuration of the ITB, a radial vane cavity (RVC) is included in the radial vane. The cavity is basically a notch cut into the radial vane which stretches from the front of the circumferential cavity to the back and slopes slightly towards the

aft of the ITB. This is shown in detail in Fig. 2b. This aids in mixing the high temperature circumferential flow with the main axial flow.

Fig. 5 shows the 3-D computational domain (with unstructured surface meshes) of the 60-degree ITB with the addition of the V-gutter. This shows the scale of the V-gutter in relation to the rest of the ITB along with its location in relation to the cavity.

A close up perspective of the V-gutter and surrounding mesh related to the SRV in the ITB is seen in Fig. 6. In this study, The V-gutter's angle of attack was varied between $\alpha = 10^\circ$ and $\alpha = -10^\circ$ with no other changes made to the geometry. By our definition, the chord of the V-gutter was parallel to the axial flow for an angle of attack of zero. The V-gutter was then rotated 10° clockwise for the positive angle of attack and rotated 10° counter-clockwise for the negative angle of attack. The mesh surrounding the V-gutter is seen where it intersects both the blade and the periodic boundary.

For the last part of this study, the V-gutter was combined with the SRV. This is shown in Fig. 7. This illustrates the V-gutter's location in relation to the SRV and its size in comparison.

5.2 Boundary Conditions

Figure 8 shows boundary conditions labeled on a baseline computational domain. Periodic boundary conditions were enforced on the two side walls 60 degrees apart. Mathematically, a periodic boundary condition states that the values calculated at one periodic boundary must be equal to the values at the same location on the matching periodic boundary. In this case, a non-conformal periodic boundary conditions was used. The difference here is that the nodes on one periodic boundary do not match the nodes on the matching periodic boundary. This results in some interpolation between the nodes in order to match the two sides as closely as

possible. Fuel was injected into the cavity through a cone injector with a half angle of 35 degree, a temperature of 290 K, a radius of 0.0014 m, a velocity of 23.18 m/s, and a mass flow of 0.0002898 kg/s per injector with a Rosin-Rammler distribution. The spray model was also set to spray the fuel in 500 separate streams. There was only one fuel injector in the 60 degree sector. Also, there was another air injector surrounding the fuel injector to allow for air-assisted fuel injection which injected air at a rate of 0.0001041 kg/s per hole. The air from the main inlet was non-vitiated and released axially through the ITB at a mass flow rate of 0.0089609 kg/s for the 60° sector which gives a mass flow of 0.0538 kg/. This resulted in a Reynolds number of approximately 14800. The main airflow constitutes roughly 80% of the total airflow while the 24 circumferential inlets and the inlet surrounding the fuel injector make up the other 15%. Each of the 24 circumferential inlets injects air at a rate of 0.0005205 kg/s for a to mass flow rate of 0.0125 kg/s. The total combined airflow for all inlets was 0.066292 kg/s. Each mass flow inlet was given a turbulent intensity of 3% and a turbulent length scale of 0.035335 m. At the outlet, the gage pressure was set to zero Pa in order to model the ITB exhausting to atmospheric air. All walls were set to no slip and with zero heat flux. The no slip condition forces zero velocity tangential to the wall at the surface, which is what causes boundary layers to develop off the walls. The zero heat flux condition forces no heat transfer between the wall and the fluid. Additional details about the operating conditions can be found in Table 1.

5.3 Convergence

For this specific problem, the solution was iterated for at least 3600 times for each case in order to reach convergence. Residual convergence levels were set to 10^{-5} for continuity, momentum, k , and ϵ , 10^{-8} for energy, and 10^{-3} for all other variables. For the first 600 iterations,

the Courant number and under relaxation terms for pressure and velocity were set at low values. After 600 iterations, all values were set back to normal (Courant Number = 15, pressure and velocity under relaxation = 0.3). The next 1000 iterations were run with first-order upwind schemes for all variables. Then for the final 2000 iterations, the second order upwind scheme was used for all variables in order to get a converged final solution for each case.

5.4 Computational Mesh

Gridgen V15.10 [24] was first used to create the unstructured surface meshes on the given geometry. The surfaces were loaded into AFLR3D [25], which was used to extrude the volume mesh from the surface meshes. The mesh was then examined to ensure that there was a minimal amount of equiangle and equivolume skew. Special attention was given to the near wall grid distribution as enhanced wall functions were utilized in this study. For each of the cases, a near wall spacing was used which resulted in y^+ values between 1 and 7, with the large majority of the values being less than 3. These values fell within the desired range of 1 to 10. The various ideal ranges of y^+ values are due to the method in which the CFD software uses in the near wall calculations. If the y^+ value lies between 1 and 10, the software will integrate all the way to the wall, but if the y^+ falls between 50 and 300, it will implement wall functions. However, if the y^+ is somewhere between these two ranges, greater error can be created because the software will not properly implement its blending functions for wall function treatment. The smaller near wall spacing was necessary to properly capture the interaction of the fluid with the surface of the flameholder and the resulting viscous boundary layer. Also, the flow was anticipated to be more complex, especially in the areas where the flameholder intersected the radial vanes. The final meshes for the V-gutter cases were all between 5.34 and 5.56 million

cells, and the mesh for the baseline case was about 2.13 million cells. The required calculations were carried out on an HP XC Linux cluster which had 2048 processors.

5.5 Results and Discussion

Figure 9 shows temperature contours at different cross-sections throughout the 3-D ITB domain with and without V-gutters. These contours at different axial locations demonstrate the trend of thermal field from inlet to outlet for each case. Fig. 6a is the result for the baseline case, where the areas of high temperature are located near the upper wall of the domain. The addition of the V-gutter (Fig. 6b) provides greater mixing of the flow in ITB. By comparing Fig. 6b and Fig. 6c it was found that increasing V-gutter angle of attack (from 0 to 10 degrees) enhances the hot-cold air mixing effect. Within the examined parameter ranges, it was also observed that the combined use of the V-gutter and the RVC in an ITB (Fig. 6d) provides the best mixing among all the cases.

The corresponding temperature contours at the center plane of the ITB are compared in Fig. 10. The center plane is in line with the injectors on the computational model and in line with the center of the V-gutters. These contours illustrate the distinct difference between the temperatures in the circumferential and the axial flows with more details. Fig. 10a shows the baseline contour, with the RVC located on the left side of the center blade. Although there is minimal mixing between the high and low temperature regions on the side opposite the RVC, there is considerable mixing between these two zones inside of the RVC. This proves that the RVC does in fact aid in mixing in ITB. The two V-gutter cases with no RVC (Figs. 10b & 10c) show no mixing on either side of the center vane, which is to be expected. The case with both

the RVC and the V-gutter (Fig. 10d) show some slight interaction between the two flow regimes on the RVC side.

The temperature contours at the exit plane are shown in Fig. 11. For the baseline configuration (Fig. 11a), the temperature contours indicate that most of the high temperature flow is located near the upper wall. Certain level of mixing between the flow leaving the circumferential cavity and the main flow has occurred, but not substantial. The V-gutter case with $\alpha = 0$ shows a much favorable temperature field, in which more hot air from the circumferential cavity has been mixed with the cold air in the axial flow. The temperature contour for the V-gutter at $\alpha = 10^\circ$ case shows very little difference compared to the $\alpha = 0$ case, except there is a slightly smaller region of maximum temperature near the upper wall and throughout the plane. For the case with the combination of the V-gutter with the RVC, the hot spot is found relatively smaller and relocated closer to the periodic plane. Although there is a larger area of more uniform temperature, strong temperature gradients are still present at the exit plane.

Fig. 12 shows a comparison of the mass averaged temperature profiles at the ITB exit plane for all the cases investigated. The mass averaged temperature values were calculated by splitting the ITB exit area into 10 annuli. The mass averaged temperature was then calculated for each annulus. It can be seen from the figure that the baseline has the worst temperature profile, which is near linear and peaked at the outer most point. For the case with the addition of the V-gutter at $\alpha = 0$, the exit temperature profile is altered - the minimum outlet temperature (at the inner wall) is raised and the maximum temperature (at the outer wall) is lowered. The case with the V-gutter positioned at $\alpha = -10$ is virtually identical to the $\alpha = 0$ case, therefore no additional analysis is presented on this case within this paper. Compared to these two cases, the

one with $\alpha = 10$ provides more improvement in the temperature profile, especially in reducing the maximum temperature at the exit. For the case with the combined V-gutter and RVC configurations, the temperature profile is much closer to parabolic with a much lower maximum temperature profile, which represents the best results among all the cases examined. An overall drop in maximum temperature of approximately 300 K was observed, which is quite significant. Future work will be devoted to verifying these results.

Fig. 13 shows the turbulence intensity levels in the ITB with different V-gutter configurations. The cutting planes shown are located on the periodic plane for each case. Fig. 13a shows the turbulence intensity predicted for the baseline configuration. Due to the interaction between the turbulent flows from main stream and that from the circumferential cavity, a zone of relatively higher turbulence intensity is generated right after the corner of the front wall of the cavity, and is continuously extended up into the cavity. It is also observed that right after the corner of the aft wall of the cavity, more turbulence is generated and transported. The high turbulence zone fills almost the entire flow passage when it reaches the exit of the ITB. From the first and second V-gutter cases, it can be seen that the V-gutter not only induces a high turbulence zone behind itself, it also results in a high turbulence spot right after the corner of the cavity's aft wall. It is also noticed that due to the obstacle of the V-gutter to the flow stream, the upper high turbulence zone within the circumferential cavity is extended further up into the cavity before it dips back down into the main flow. This phenomenon would cause the increase in the amount of mass transfer or entrainment into and out of the upper circumferential cavity. Fig. 13c also shows that by giving the V-gutter a positive angle of attack ($\alpha = 10$), the area of high turbulence generated by the V-gutter is directed down towards the lower wall of the ITB. For the case with both a V-gutter and RVC (Fig. 13d), the most widespread elevated turbulence

levels downstream of the V-gutter are observed. The high turbulence levels also extend all the way into the upper cavity, as well as to the areas immediately downstream of the aft wall of the cavity. Also, a zone of very high turbulence appears just upstream of the exit plane.

Along with turbulence intensity, the addition of the V-gutter also alters the vorticity within the ITB. This can be seen in Figs. 14a – 14d. The baseline configuration (Fig. 14a) shows areas of high vorticity propagating from the front and back corners of the circumferential cavity. The high vorticity areas near the upper wall of the cavity are induced primarily by the jet flows from air injections. In addition to increase the vorticity levels, the inclusion of the V-gutter (Fig. 14b) also pushes the area of high vorticity further into the circumferential cavity. Similar to turbulence transport, the vorticity transport also assists in the mass transfer and entrainment into and out of the main and circumferential streams. Beside the high vorticity area right behind the V-gutter, the vortices shedding from the trailing edges of the V-gutter are clearly depicted in the figures. At a positive V-gutter angle of attack (Fig. 14c), the areas of high vorticity around the V-gutter are more contained to the wall of center body in the ITB. In the case with both a V-gutter and an RVC (Fig. 14d), the high vorticity zone induced by the front corner of the cavity is broken into two areas due to the mass transfer and mixing caused by the RVC. As a result of such vorticity change, the overall vorticity levels near the ITB exit plane are reduced.

The inclusion of V-gutter could cause additional pressure drop across the ITB. Fig. 15 shows the comparison of the relative pressure drops (DP) associated with each ITB configuration, which were calculated using the following formula:

$$DP = \frac{P_{in} - P_{out}}{P_{in}} \times 100\% \quad (31)$$

As seen from Fig. 15, the pressure drop for each case is very low. Compared to the baseline case, cases with the inclusion of the V-gutter add very small to negligible pressure drop. The effects of the angle of attach of the V-gutter on the pressure drop is also almost negligible. The highest pressure drop occurs in the case with both a V-gutter and an RVC. This is reasonable, as the radial cavity could induce flow separation and turbulent mixing, resulting in additional pressure loss inside the ITB. Other design modifications for the V-gutter are being considered in order to minimize the higher pressure drop incurred from the combination of the V-gutter and the RVC.

One of the critical components of the flow structure of the ITB is the high G swirling flow within the circumferential cavity. This is what allows for combustion to occur in such a small space and at such a high efficiency. Fig. 16 shows the tangential acceleration within the circumferential cavity at different constant axial locations. The tangential acceleration was calculated by:

$$a_t = \frac{v_t^2}{r} \quad (32)$$

where a_t and v_t are the tangential acceleration and velocity, respectively. When this figure is examined, it can be seen that for all axial locations, the tangential acceleration is at its minimum values near the upper and lower bounds of the cavity. It can also be seen that the values are higher near the front of the cavity, drop near the center of the cavity, and increase again near the aft of the cavity. This is to be expected since there are angled air injectors up and down stream of the fuel injector, which injects normal to the wall. This effectively counteracts the tangential acceleration at the center of the cavity, but it is quickly increased by the angled air injectors

before and after the fuel injector. The g-load within the cavity can be calculated by dividing these tangential acceleration numbers by the gravity constant.

$$G = a_t / g \quad (33)$$

where g represents the gravity constant (9.81 m/s²).

Since the main purpose of the V-gutter within the ITB is to attempt to mix the combusted products with the main axial flow, it is important to determine whether the amount of mass exchanged between the circumferential and main cavities changes with the inclusion of a V-gutter. This was visualized in Fig. 17 and Fig. 18. Fig. 17 explains in detail the geometric details of the interfaces between the circumferential and main cavities. It shows that the upstream and downstream walls of the circumferential cavity are actually at different heights, which results in a small area on the downstream side of the cavity where combustion products can escape from the circumferential cavity in the x-direction. Therefore, it was of interest to know the ratio of mass escaping across the axial and radial interface. This was shown in Fig. 18, where it can be seen that a significantly larger percentage of the mass flow leaving the circumferential cavity exits across the tangential interface. This indicates that the combustion products are transported from the circumferential cavity more quickly with the inclusion of the V-gutter, which allows for more mixing time with the main flow. This is consistent with the exit temperature profile findings.

VI. V-Gutter Parametric Study

6.1 Physical Model

The setup used for this analysis exactly duplicated the experimental results mentioned above. A two-dimensional approximation was made by examining findings from these experimental results. The same approximation was also made by the author of the experimental paper. Fig. 19 shows the full domain, which included the V-gutter within a cavity. The full dimensions of the test section were 305mm x 50mm. The shape used for the V-gutter was an equilateral triangle with each side equal to 25mm. This resulted in a blockage ratio of 0.5 within the cavity. Flow into the channel was a premixed combination of propane-air with an equivalence ratio of 0.6. It was given a velocity of 10 m/s and initially had 2% turbulent intensity.

For the parametric study, the inlet turbulent intensity was varied to determine the effects on downstream mixing. After the initial intensity of 2%, additional cases of 25%, 50%, 75% and 100% were modeled. The angle of attack of the V-gutter was also varied. In this case, the angle of attack was defined by the included angle between the axial direction of the cavity and the chord of the V-gutter. Initially this angle was zero. It was then rotated to 5°, 10°, 20°, and 30°. The definition used in this paper was that clockwise rotation indicated a positive angle.

6.2 Boundary Conditions

The initial boundary conditions were chosen to match those of the experimental setup. As mentioned earlier, the inlet had a velocity of 10 m/s, 2% turbulent intensity, and a hydraulic diameter of 0.05 m. This resulted in a Reynolds number of about 35200. It was also 300 K. The air from the main inlet was non-vitiated and released axially through the ITB. The equivalence

ratio of 0.6 resulted in a propane mass fraction of 0.037, an oxygen mass fraction of 0.2245, and a nitrogen mass fraction of 0.7385. All of the walls, including the V-gutter itself, were assumed to be adiabatic. The outlet was set as an outflow boundary for the validation study since the experiment was run in a closed wind tunnel, and then it was changed to a pressure outlet with gauge pressure set to zero for the parametric study to simulate a combustor exhausting to the atmosphere.

6.3 Convergence

For this problem, the 3000 iterations were performed for each case in order to reach convergence. Residual convergence levels were set to 10^{-5} for continuity, momentum, k , and ϵ , 10^{-8} for energy, and 10^{-3} for all other variables. For the first 1500 iterations, standard pressure discretization was used along with first order discretization for all other variables. After this was completed, the pressure discretization scheme was changed to PRESTO! And second order discretization for all other variables was implemented for the final 1500 iterations. Convergence was further verified by monitoring key variables such as temperature and velocity.

6.4 Computational Mesh

For this paper, a grid independent study was performed on the V-gutter problem in order to determine the optimal grid which performs with a good balance of accuracy and computational speed. Mesh sizes ranged from approximately 21,000 cells to 361,000 cells, and all meshes were structured. Gridgen V 15.10 [24] was used to create all the meshes. For each case, the near wall spacing was held constant which resulted in y^+ values no higher than four at any point along the wall and V-gutter. This was important due to that fact that enhanced wall

functions were used. This required that the y^+ values not exceed ten for results of the highest accuracy. The smaller near wall spacing was necessary to properly capture the interaction of the fluid with the surface of the flame holder and the resulting viscous boundary layer. The meshes were also examined for skewness and area ratio to ensure quality results.

The parameters used to determine the grid convergence were the area averaged exit temperature and the area averaged axial velocity. Table 2 shows each mesh used and the results obtained from the mesh. After examining the results of the grid independent study, it was determined that the optimal grid size was approximately 279,000 cells. This mesh provided converged results without exceptionally high computational times. The required calculations were carried out on a Linux cluster with 68 processors containing 208 cores.

6.5 Results and Discussion

The numerical results were first compared to experimental results in order to confirm validity. With the turbulence and combustion models used, extremely accurate results were not expected. Instead, fairly accurate trends in data were obtained. Fig. 20 shows the results from this validation study. The data shown was turbulent kinetic energy, and the locations are from the V-Gutter centerline and the V-Gutter Y-max extended axially towards the outlet. The x parameter is normalized axial distance which is normalized by the V-gutter size (25 mm).

The specifications used in the experimental setup were used as the baseline case for the parametric analysis. Fig. 21a shows contours of turbulent intensity within the baseline domain. This will aid in showing the magnitude, and to some extent the size, of the mixing area downstream of the V-gutter. Fig. 21b shows a magnified view of the velocity vectors directly downstream of the V-gutter. This shows the exact size and shape of the recirculation zone. The

vectors were colored by axial velocity. These results were compared to cases with varying turbulent intensity and angle of attack.

Fig. 22 shows a comparison of the turbulent intensity contours for all four of the different inlet turbulent intensities. It can be seen from the figures that increasing the inlet turbulent intensity greatly increases the turbulent intensity downstream of the V-gutter. As the inlet turbulent intensity is raised, the turbulent eddies shedding off of the trailing edges become larger in both size and magnitude. Also, the distance downstream of the V-gutter in which the turbulent intensity is elevated increases for each of the first three cases. It appears that no change occurs between the 75% and 100% case, which may indicate that there is a limit on the effect of raising the inlet turbulent intensity.

Fig. 23 compares the turbulent intensity contours for both of the increased angles of attack. This does not produce as dramatic of results as the increase in inlet turbulent intensity, but there are noticeable differences. It can be seen that the areas of high turbulent intensity shedding off of the two trailing edges remain separate instead of merging into one zone, which is what occurs in the baseline case. As the angle is increased the turbulent intensity created by the upper trailing edge seems to increase. There is also more turbulence shedding from the lower wall of the cavity as the angle is increased.

The combination of changes in inlet turbulent intensity and angle of attack was then shown in Fig. 24. This created some interesting differences in the turbulent intensity contours. With an angle of attack of 20° , the area of very high turbulent intensity directly downstream of the V-gutter was much larger than that caused by only an increase in inlet turbulent intensity. On the upper wall of the cavity downstream of the V-gutter, there were slightly lower turbulent intensity levels but a larger area of elevated levels. Also, the lower cavity wall created much

higher turbulent intensity levels which eventually merged with that coming off of the lower trailing edge, but the area of elevated turbulent intensity was smaller than that of zero angle of attack.

The effect of these parameters on the recirculation zone was also studied. Fig. 25 shows the velocity vectors directly downstream of the V-gutter for each of the inlet turbulent intensity cases. These results show what would be expected after seeing the turbulent intensity contours. As the inlet turbulent intensity is increased, the velocities downstream of the V-gutter are increased. This includes the recirculation velocity. The size of the recirculation areas, however, seems to remain the same.

When the angle of attack was varied, a different effect was observed. Fig. 26 shows the velocity vector plots for the different angles of attack. This time the velocities remained basically unchanged, but the size and location of the recirculation zones changed. As the V-gutter was rotated further in the clockwise direction, the lower recirculation zone increased in size while the upper decreased.

The combination of varying angle of attack and turbulent intensity also gave results that would be expected. The velocity throughout the domain increased and the lower recirculation zone was enlarged, but both effects were lessened compared to when each was varied individually. This was shown in Fig. 27.

Once these comparisons were complete, it was of interest to determine what effect these observed results would have on the combustion properties of the V-gutter. Fig. 28 compares the temperature contours of the baseline setup with the most extreme case from the inlet turbulent intensity study, the angle of attack study, and the combined study. Fig. 29 gives the same comparison as Fig. 28, except using carbon dioxide levels instead of temperature. It can be seen

that both angle of attack and inlet turbulent intensity affect the rate of combustion. When the inlet turbulent intensity alone is increased, the CO₂ levels show that complete combustion occurs directly downstream of the V-gutter, which is much different than the baseline case. This can also be seen in the temperature contour, where low temperature flow does not penetrate nearly as far downstream when compared to the baseline. When the angle of attack alone is increased, the amount of combusted reactants seems to increase below the V-gutter but decreased above the V-gutter. When both high angle of attack and high inlet turbulent intensity are combined, all reactants are combusted the earliest within the domain when compared to all other cases. As would be expected from the high angle of attack scenario, complete combustion occurs sooner below the V-gutter than above. However, both are much shorter than the baseline scenario.

Table 3 gives a comparison of each of the four cases examined in the previous two figures. Maximum values of temperature, axial velocity, and turbulent intensity are given. For each of these three parameters, the baseline configuration produces the minimum of all the cases. Increasing only the inlet turbulent intensity causes a large increase in all three values. The velocity and turbulence values approximately double while the temperature rises by approximately 65 K. When only angle of attack is increased, all of the values are raised as well, but not to the same level as what is achieved by raising the inlet turbulent intensity. The combination of increasing both angle of attack and inlet turbulent intensity results in the highest values of both turbulent intensity and axial velocity. The temperature is approximately the same as increasing turbulent intensity alone.

These maximum values were further studied in Fig. 30, which shows every combination of angle of attack and inlet turbulent intensity. This shows that any increase in angle of attack or turbulent intensity results in an increase in the maximum values being studied. There are some

differences in how these are increased, though. While axial velocity and turbulent intensity increase fairly uniformly as inlet turbulent intensity is increased for each angle of attack, it appears that an increase in inlet turbulent intensity has less effect on the maximum temperature as the angle of attack is increased.

Table 4 studied the changes in size of the recirculation zone. These sizes were determined by plotting streamlines and measuring the x and y maximum and minimum locations of recirculation. Since two separate recirculation areas occur behind the V-gutter, a separate top and bottom y span of recirculation was specified. The table showed that for an angle of attack of zero, there was basically no change in recirculation zone size as inlet turbulent intensity was increased. Increasing the angle of attack resulted in a decrease in the x span and an increase in the total y span of the recirculation zone. Also, the lower recirculation zone became larger while the upper recirculation zone shrank. As inlet turbulent intensity was increased, there was a small decrease in x span and no change in the total y span, but there were significant differences in the y spans of the upper and lower recirculation zones. Increases in the turbulent intensity resulted in a larger lower recirculation zone and a smaller upper recirculation zone.

Another major consideration when looking at a V-gutter design is the pressure loss associated with putting this kind of blockage in the flow. For this case, the friction factor was used to represent the pressure drop [26]. The formula used to calculate the friction factor was as follows:

$$f = 2\rho D_h \left(\frac{\Delta p}{\Delta x} \right) \left(\frac{A_c}{\dot{m}} \right)^2 \quad (32)$$

Fig. 31 shows the effect that varying both inlet turbulent intensity and angle of attack have on the friction factor. It can be seen from the results that the friction factor decreases significantly as

turbulent intensity decreases. Also, the initial change in angle from 0° to 5° causes a drop in friction factor, but any increase in angle beyond this for the range in question makes little difference. In order to further minimize the friction factor, and thus minimize the pressure drop, the interior leading edge angle of the V-Gutter could be decreased in order to create a smaller blockage ratio, but this variation of parameters was beyond the scope of this study.

Another important parameter that was examined was the wall temperature profile. It was of interest to determine what effect the variation in inlet turbulent intensity and angle of attack would have on these temperature profiles. Fig. 32 shows the temperature profiles of the upper and lower cavity walls for both $\alpha = 0^\circ$ and $\alpha = 30^\circ$. From these, it can be seen that at $\alpha = 0^\circ$, an increase in inlet turbulent intensity results in the wall reaching high temperatures closer to the inlet. This implies that combustion occurs closer to the inlet along the walls. There was no noticeable difference between the upper and lower walls at this angle of attack, which was expected. At $\alpha = 30^\circ$, the main difference noticed was a dip in wall temperature on the lower wall just downstream of the v-gutter. This is caused by the v-gutter forcing the uncombusted mixture towards the lower wall at the increased angle. The effect vanishes at high inlet turbulent intensities because combustion of the reactants occurs closer to the inlet.

Fig 33 shows the temperature profiles of the upper and lower v-gutter walls for both $\alpha = 0^\circ$ and $\alpha = 30^\circ$. Once again, an increase in turbulent intensity resulted in high wall temperatures closer to the leading edge of the v-gutter at $\alpha = 0^\circ$. When the angle of attack was increased to $\alpha = 30^\circ$, the wall temperature reached high levels much closer to the leading edge on the upper v-gutter wall. However, the opposite effect was observed along the lower wall. Also, these cases show a decrease in wall temperature near the trailing edge of the v-gutter where the zero angle of

attack cases showed no decrease. This becomes less noticeable as the inlet turbulent intensity is increased. The last wall compared was the rear v-gutter wall.

Fig. 34 shows the temperature profiles of the rear v-gutter wall at $\alpha = 0^\circ$ and $\alpha = 30^\circ$. For $\alpha = 0^\circ$, the temperature profiles are basically constant across the entire wall with this constant value increasing as the inlet turbulent intensity increases. When the angle of attack was increased to $\alpha = 30^\circ$, there was an initial high spike of temperature near the upper edge of the wall. Further along the wall, the temperature slowly increased with a small drop-off at the lower edge. It was also noticed at both angles of attack that an increase in turbulent intensity resulted in an increase of temperature everywhere along the wall.

VII. Conclusion

7.1 ITB Study

A 3-D CFD analysis was performed to study the turbulent flow and combustion in an ITB with the inclusion of a V-gutter flameholder at different angle of attack. Numerical simulations were conducted for Jet-A spray combustion using a Realizable k- ϵ turbulence model and an eddy dissipation combustion model. The CFD results clearly showed the functionality of the important ITB features such as the SRV, the radial vane cavity, the high-g circumferential cavity, and the V-gutters.

Within the investigated parameter ranges, it was found that the addition of the V-gutter did in fact improve the temperature distribution at the exit of the ITB.

The vortices induced by the V-gutter resulted in larger zones of high turbulence, resulted in enhanced mixing and entrainment between the circumferential flow with combustion products and the main axial flows of air in the ITB.

The improvements in mixing and exit profile were produced with little expense in pressure drop. A combined V-gutter and radial vane cavity was found to provide the most improvement in ITB performance.

Future studies include the investigation of alternate placements of the V-gutter for ITB combustion performance improvement and geometrical optimization at different operating conditions.

7.2 V-Gutter Parametric Study

A parametric analysis was performed to study the effects of varying angle of attack and inlet turbulent intensity on the mixing effects of a V-gutter and how this affected combustion. Numerical simulations were conducted for propane-air combustion using a Realizable $k-\epsilon$ turbulence model and an eddy-dissipation combustion model. In doing this, a grid independent study and a validation study were also performed.

Results showed that an increase in inlet turbulent intensity results in a higher level of turbulence downstream of the V-gutter which in turn causes an increase in velocity, especially the recirculated negative axial velocity. When the angle of attack was increased, the lower recirculation zone grew while the other shrunk. The turbulence levels remained relatively constant, though. When both inlet turbulent intensity and angle of attack were increased, both of the previously mentioned effects were observed, but each had lesser impact compared to when each parameter was varied separately. Also, an increase in turbulent intensity resulted in high friction factors and higher wall temperatures with steeper increases in temperature profiles.

7.3 Final Conclusions

It has been shown that the inclusion of a v-gutter bluff body flameholder results in significant improvement of the downstream mixing of combusted products in the ITB. Using the results of the first and second studies in this paper, it can be seen that the amount of benefit gained from the v-gutter and the location of greatest benefit depend upon factors such as inlet conditions and angle of attack of the v-gutter. These results show that the v-gutter's effect can be changed to suite different ITB configurations. A variety of different ITB geometries could benefit from the inclusion of a v-gutter to mix the combusted products with the main flow.

Someone would just need to determine where mixing is most needed in their domain and then use the appropriate v-gutter configuration. However, more work needs to be done to determine what other property or geometric changes could alter the performance of the v-gutter as well.

References

References

- [1] Cengel, Y.A., Boles, M.A., 2002, Thermodynamics: An Engineering Approach, McGraw Hill.
- [2] Hill, P., Peterson, C., 1992, Mechanics of Thermodynamics of Propulsion, Addison Wesley Longman.
- [3] Sirignano, W.A., Delplanque, J.P., et al, 1997, "Selected Challenges in Jet and Rocket engine Combustion Research," 33rd AIAA/ASMESAE/ASEE Joint propulsion Conference, Seattle, WA , Paper No. AIAA-97-2701.
- [4] Sirignano, W.A., and Liu, F., 1999, "Performance Increases for Gas-Turbine Engines Through Combustion Inside the Turbine," Journal of Propulsion and Power, Vol. 15, No. 1, pp. 111-118.
- [5] Liu, F., and Sirignano, W.A., 2000, "Turbojet and Turbofan Engine Performance Increases Through Turbine Burners," 38th Aerospace Sciences Meeting & Exhibit, Reno, NV, Paper No. AIAA-2000-0741.
- [6] Mawid, M.A., Park, T.W., Thornburg, H., Sekar, B., and Zelina, J., 2005, "Numerical Analysis of Inter-Turbine Burner (ITB) Concepts for Improved Gas Turbine Engine Performance," 43rd AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, Paper No. AIAA-2005-1162.
- [7] Zelina, J., Ehret, J., Hancock, R. D., Shouse, D. T., and Roquemore, W. M., 2002, "Ultra-Compact Combustion Technology Using High Swirl for Enhanced Burning Rate", 38th Joint Propulsion Conference & Exhibit, Indianapolis, IN, Paper No. AIAA -2002-3725.
- [8] Zelina, J., Shouse, T., and Hancock, R. D., 2004, "Ultra-Compact Combustors for Advanced Gas Turbine Engines," ASME IGTI Paper 2004-GT-53155, Vienna, Austria.

- [9] Zelina, J., Sturgess, G. J., Mansour, A., and Hancock, R. D., 2003, "Fuel Injection Design for Ultra-Compact Combustor," Int. Symp. Air Breathing Engines Paper No. ISABE-2003-1089.
- [10] Zelina, J., Sturgess, G. J., and Shouse, D. T., 2004, "The Behavior of an Ultra-Compact Combustor (UCC) Based on Centrifugally-Enhanced Turbulent Burning Rates," 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conf., Fort Lauderdale, FL, AIAA Paper 2004-3541.
- [11] Lin, C., Sekar, B., Holder, R., Zelina, J., Thornburg, H., 2008, "Numerical Simulation of Inter-Turbine Burner (ITB) Flows with the Inclusion of V-Gutter Flame Holders," 2008 Gas Turbine Technical Congress and Exposition, Berlin, Germany, GT2008-50337.
- [12] White, F., 2006. Viscous Fluid Flow, McGraw Hill.
- [13] Thornburg, H., Sekar, B., Zelina, J., Greenwood, R., and Skudarnov, P., 2007, "Geometrical Parametric Studies of Inter-Turbine Burner (ITB) for Improved Performance," 43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conf., Cincinnati, OH, AIAA Paper 2007-5099.
- [14] Mawid, M. A., Thornburg, H., Sekar, B., and Zelina, J., 2006, "Performance of an Inter-Turbine Burner (ITB) Concept with Three Different Vane Cavity Shapes," 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conf., Sacramento, CA, AIAA Paper 2006-4740.
- [15] Thornburg, H., Sekar, B., Zelina, J., and Greenwood, R., 2007, "Numerical Study of an Inter-Turbine Burner (ITB) Concept with Curved Radial Vane," 45th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, AIAA Paper 2007-649.
- [16] Thornburg, H., Sekar, B., Zelina, J., Lin, C., Holder, R., 2008, "Prediction of Inter-Turbine Burner (ITB) Performance with Curved Radial Vane Cavity at Various Equivalence

- Ratios,” ASME Turbo Expo 2008: Power for Land, Sea, and Air, Berlin Germany, GT2008-50192.
- [17] Fujii, S., Gomi, M., Eguchi, K., 1978, “Cold Flow Tests of a Bluff-Body Flame Stabilizer,” *Journal of Fluids Engineering*, Vol. 100.
- [18] Sanquer, S., Bruel, P., Deshaies, B., 1998, “Some Specific Characteristics of Turbulence in the Reactive Wakes of Bluff Bodies,” *AIAA Journal*, Vol. 36 No. 6.
- [19] Fujii, S., Eguchi, K., 1981, “A Comparison of Cold and Reacting Flows Around a Bluff-Body Flame Stabilizer,” *Journal of Fluids Engineering*, Vol. 103.
- [20] Kim, W., Lienau, J., Slooten, P., Colket, M., Malecki, R., Syed, S., 2006, “Towards Modeling Lean Blow Out in Gas Turbine Flameholder Applications,” *Journal of Engineering for Gas Turbines and Power*, Vol. 128.
- [21] Fureby, C., 1995, “Large-Eddy Simulation of Turbulent Anisochoric Flows,” *AIAA Journal*, Vol. 33.
- [22] Porumbel, I., Menon, S., 2006, “Large Eddy Simulation of Bluff Body Stabilized Premixed Flame,” *AIAA Aerospace Sciences Meeting and Exhibit*, Reno, NV, AIAA 2006-152.
- [23] Fluent Inc., FLUENT 6.3.26, www.fluent.com, 2007
- [24] Pointwise, Inc., GRIDGEN V15, 2004.
- [25] AFLR3D, www.erc.msstate.edu
- [26] Ahn, H. S., 2007, “Heat Transfer Enhancement for Turbulent Flow Through Blockages With Round and Elongated Holes in a Rectangular Channel,” *Journal of Heat Transfer*, Vol. 129.

Appendices

Appendix I. ITB Figures and Tables

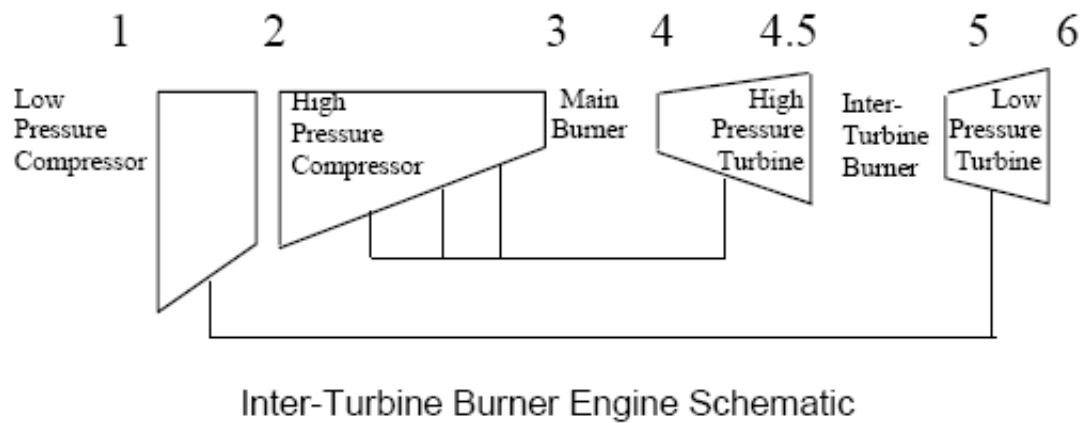


Figure 1 Schematic of ITB Placement Within a Turbojet Engine

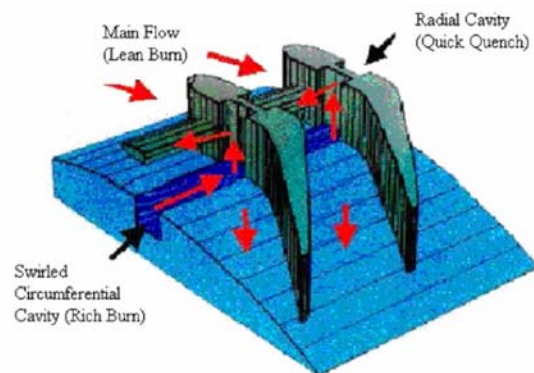


Figure 2 UCC Design Concept

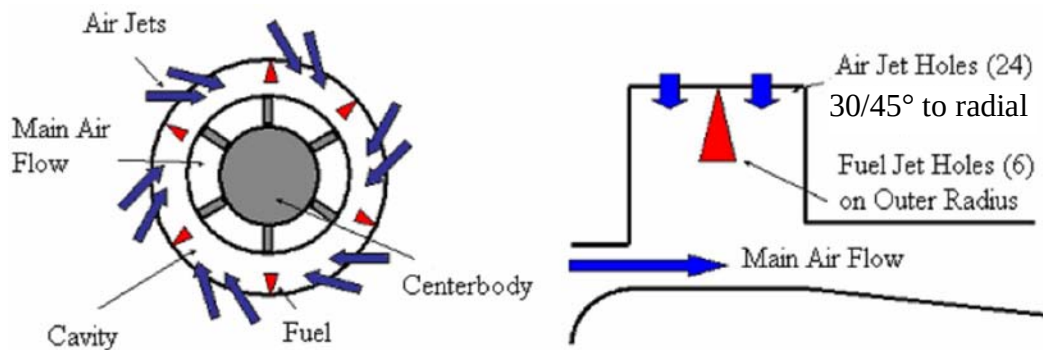
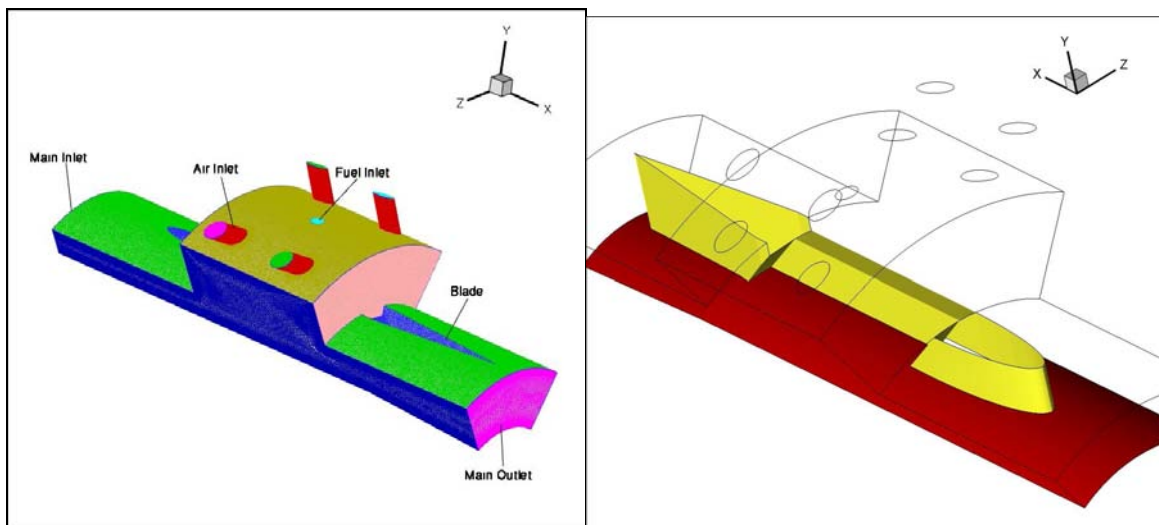


Figure 3 Diagrams illustrating the air inlet locations and directions along with fuel injector positioning.



a. Baseline SRV Setup Exterior

b. SRV Detail

Figure 4 A 60 degree cross-section of the baseline configuration ITB.

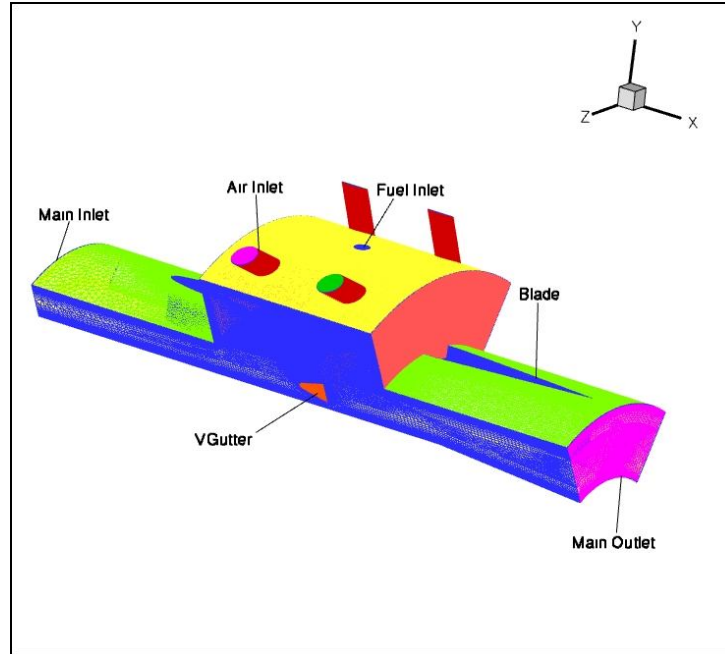
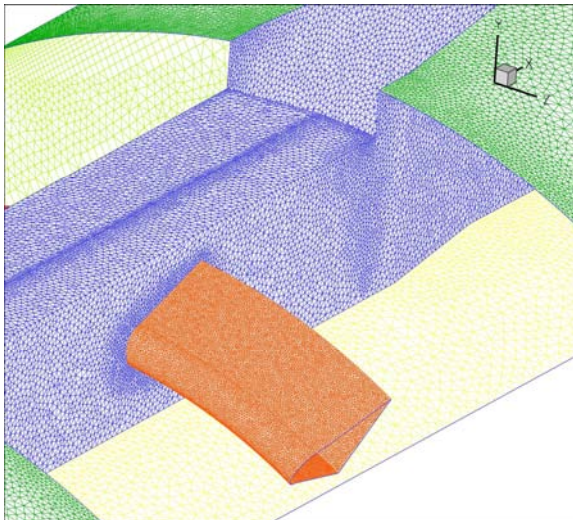
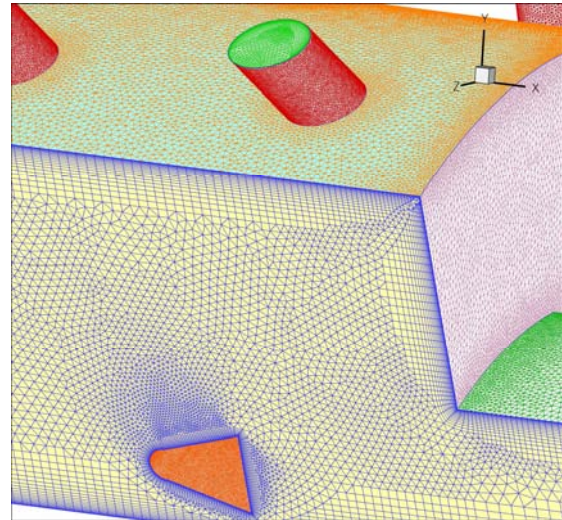


Figure 5 The 60-degree section ITB with the addition of the V-gutter.



a. V-Gutter Intersected with Blade



b. V-Gutter Intersected with Periodic Plane

Figure 6 A close-up view of the V-gutter flameholder and the surrounding mesh.

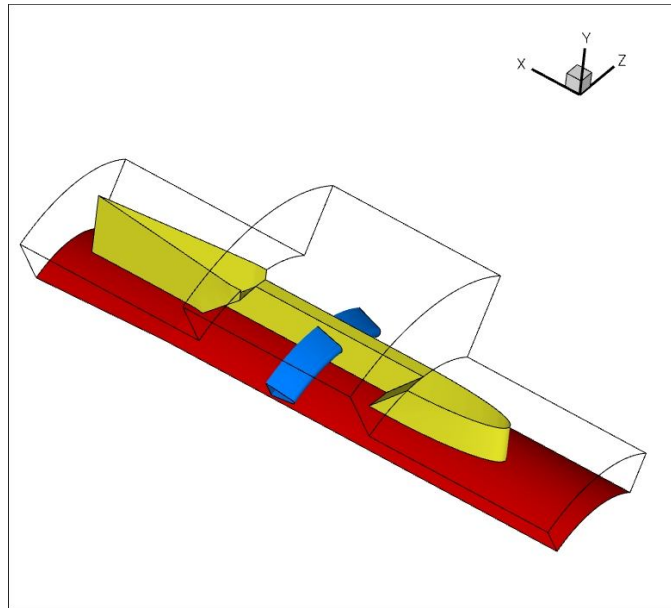


Figure 7 Detailed View of the V-Gutter with the SRV

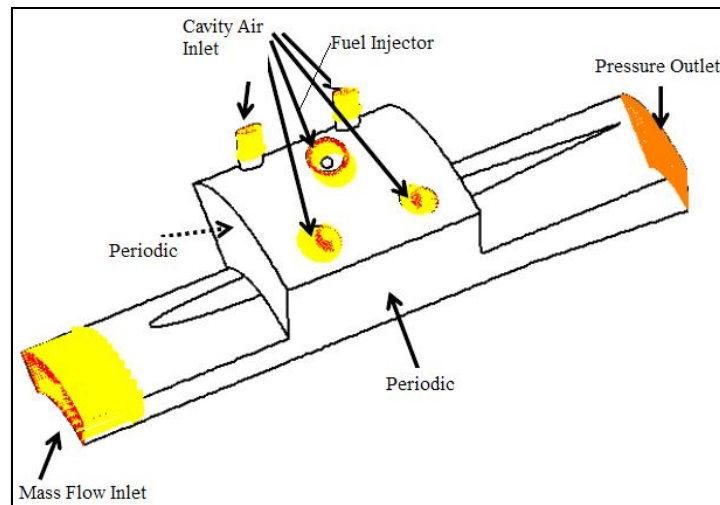
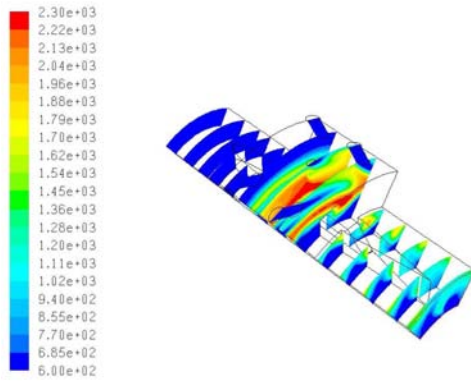


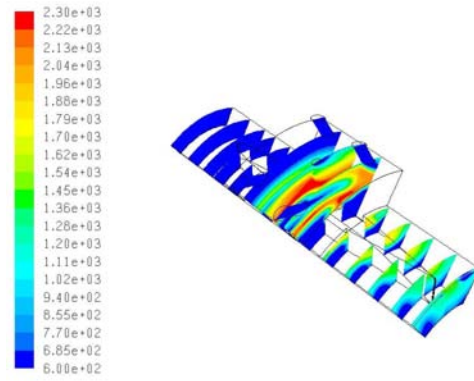
Figure 8 Baseline ITB with labeled boundary conditions.

Table 1 Baseline ITB Boundary Flow Conditions

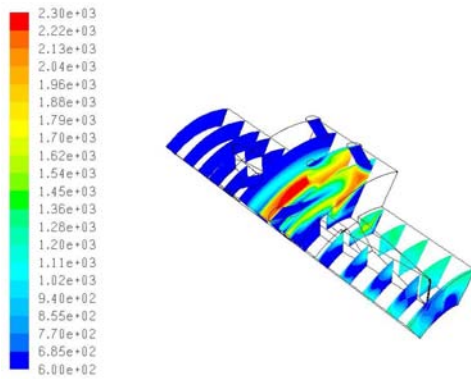
Operating Pressure (atm)	Main Inlet Temp. (K)	Main Inlet Airflow Rate (kg/s)	Main Inlet Turbulence Intensity (%)	Cavity Airflow Inlet Temp. (K)	Cavity Airflow Rate per Hole (kg/s)	Cavity Fuel Flow Rate per Injector (kg/s)	Cavity Fuel Temp. (K)	Total Airflow Rate (kg/s)	Overall Equivalence Ratio
1.0	533	8.961×10^{-3}	3.0	533.15	5.205×10^{-4}	1.081×10^{-4}	290	2.186×10^{-3}	0.142



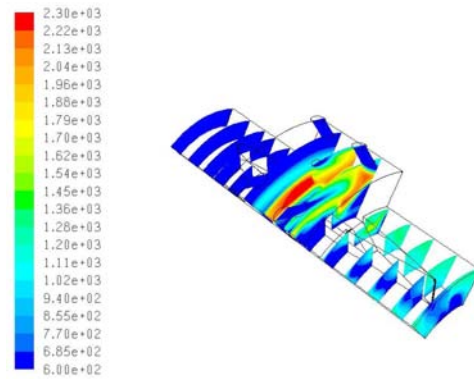
a. Baseline



b. V-gutter $\alpha = 0$



c. V-gutter $\alpha = 10$



d. V-gutter + RVC

Figure 9 Axial slices of temperature (K) contours throughout the ITB domain.

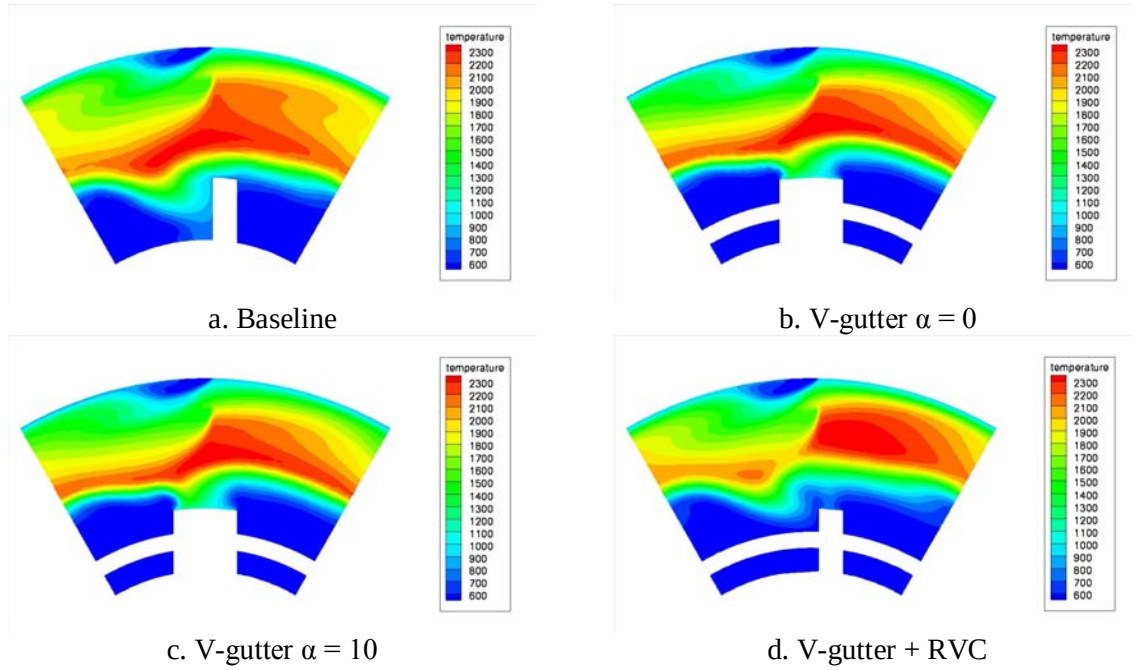


Figure 10 Temperature (K) contours at the center plane inline with both the injector and the center of the V-gutter.

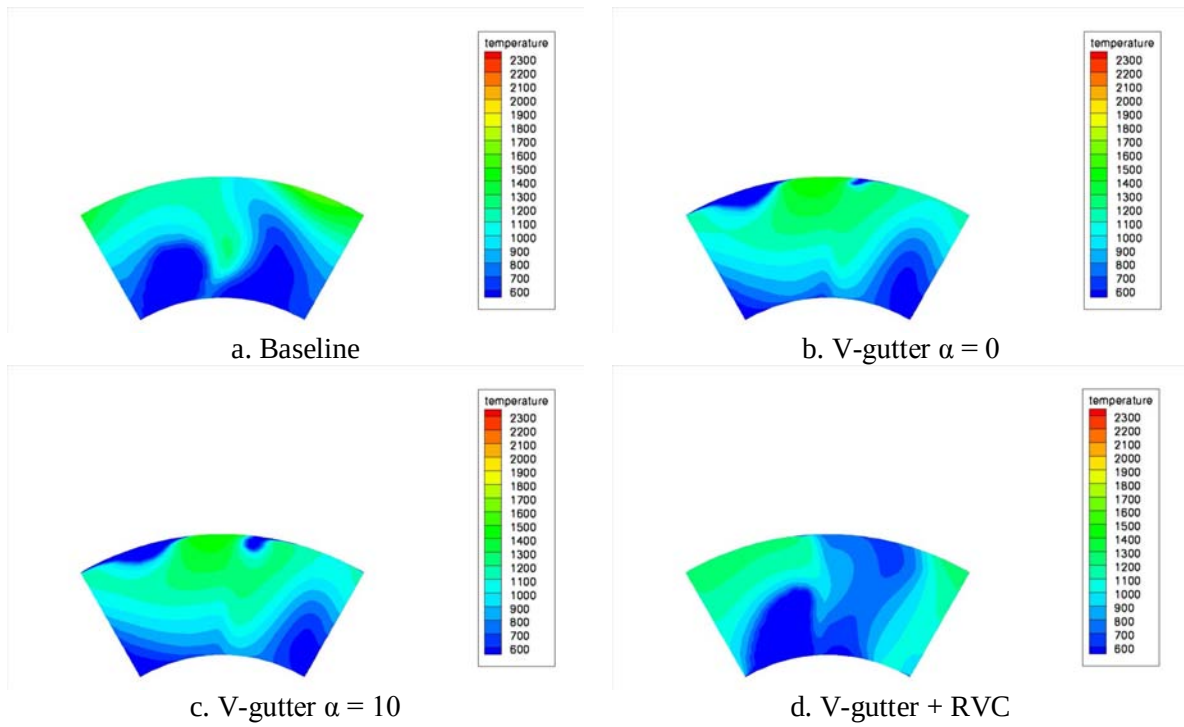


Figure 11 Temperature (K) contours at exit plane of ITB.

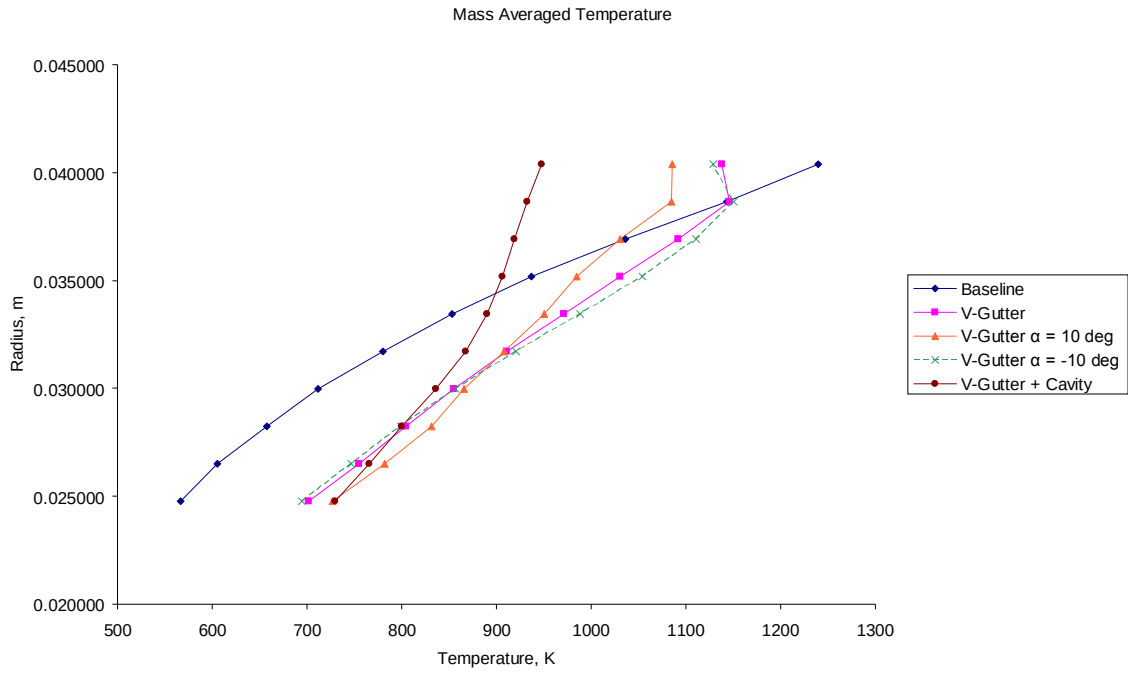


Figure 12 Comparison of the mass averaged temperature profiles on the exit plane of ITB.

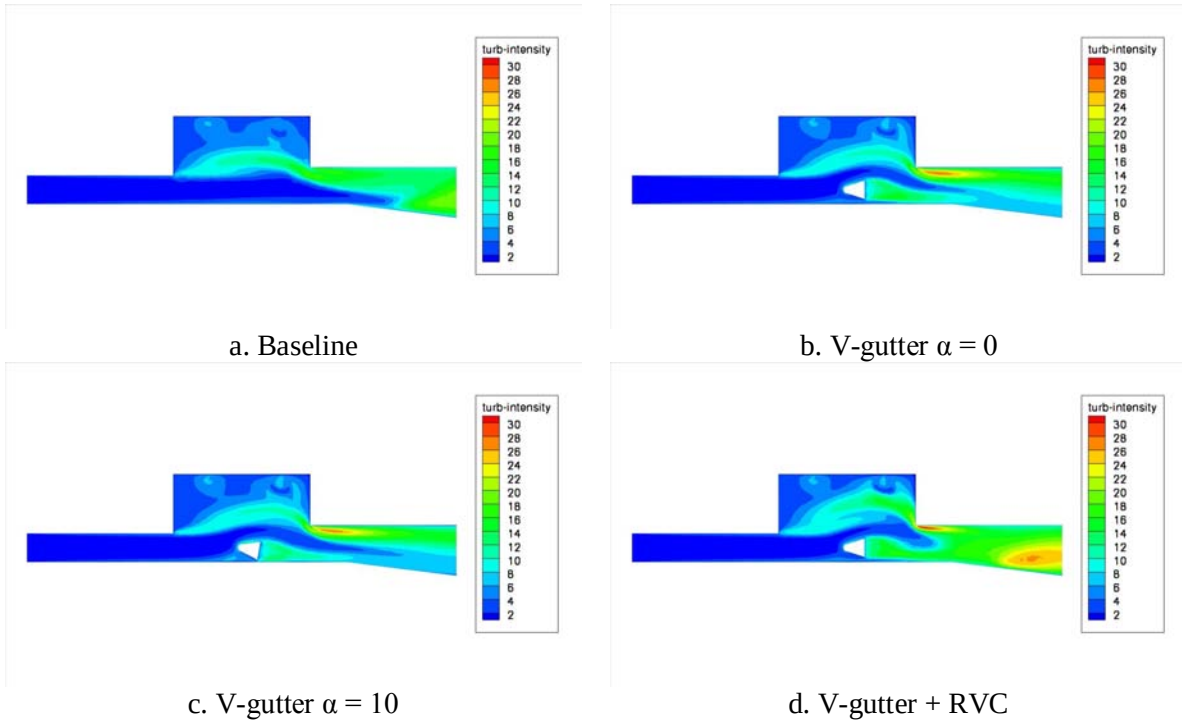


Figure 13 Turbulent intensity contours on the periodic plane of the computational model.

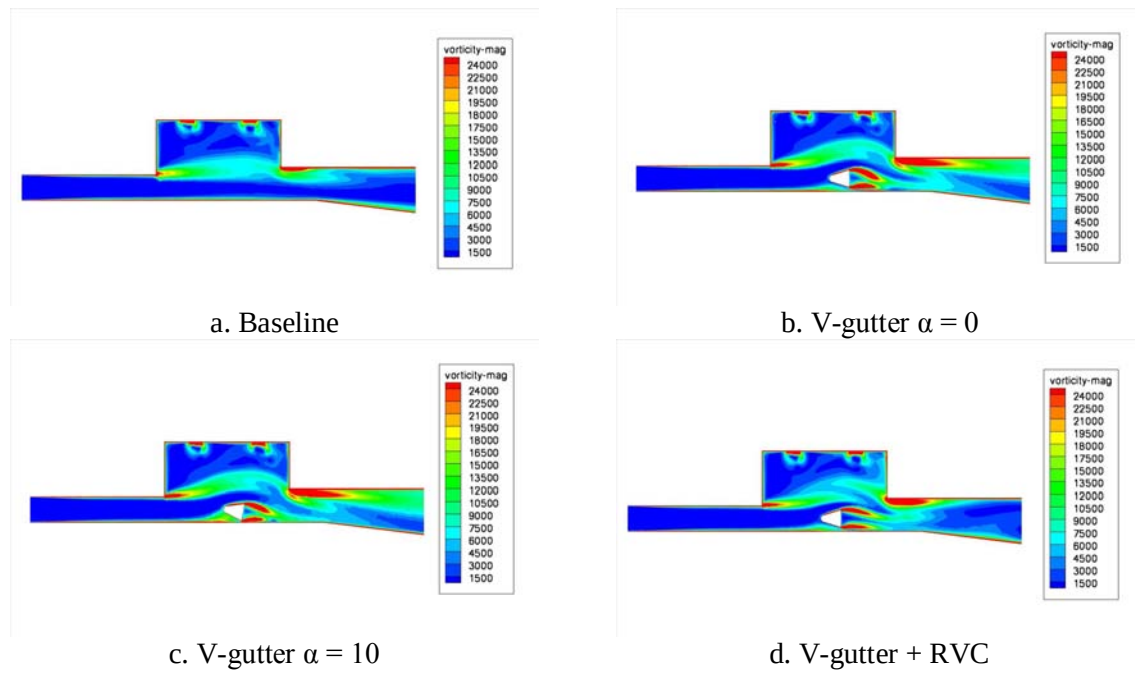


Figure 14 Contours of vorticity on the periodic plane of the computational model.

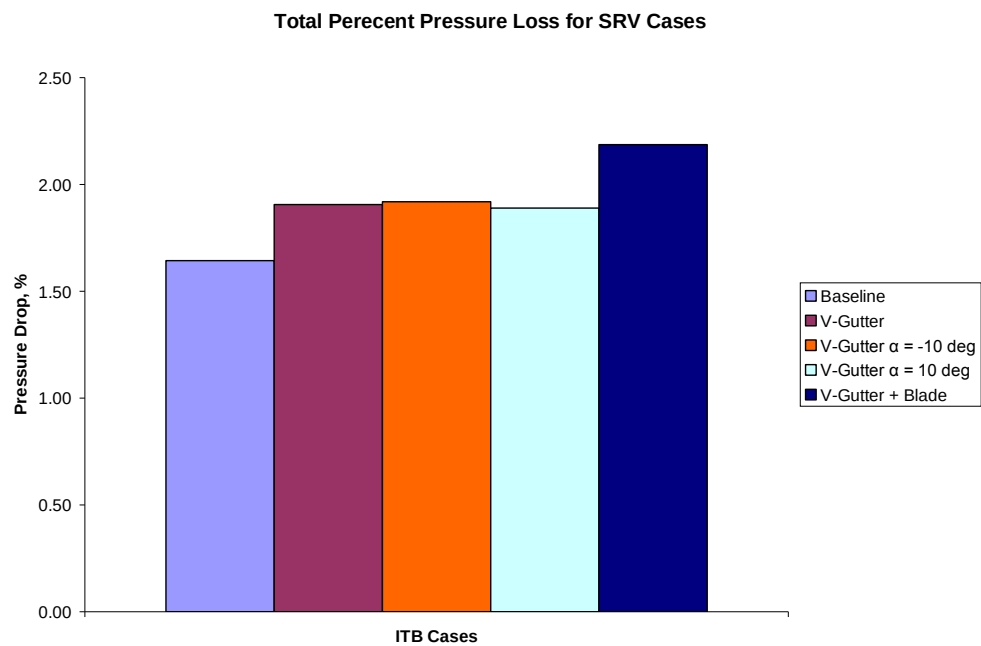


Figure 15 Comparison of relative pressure drop across the ITB

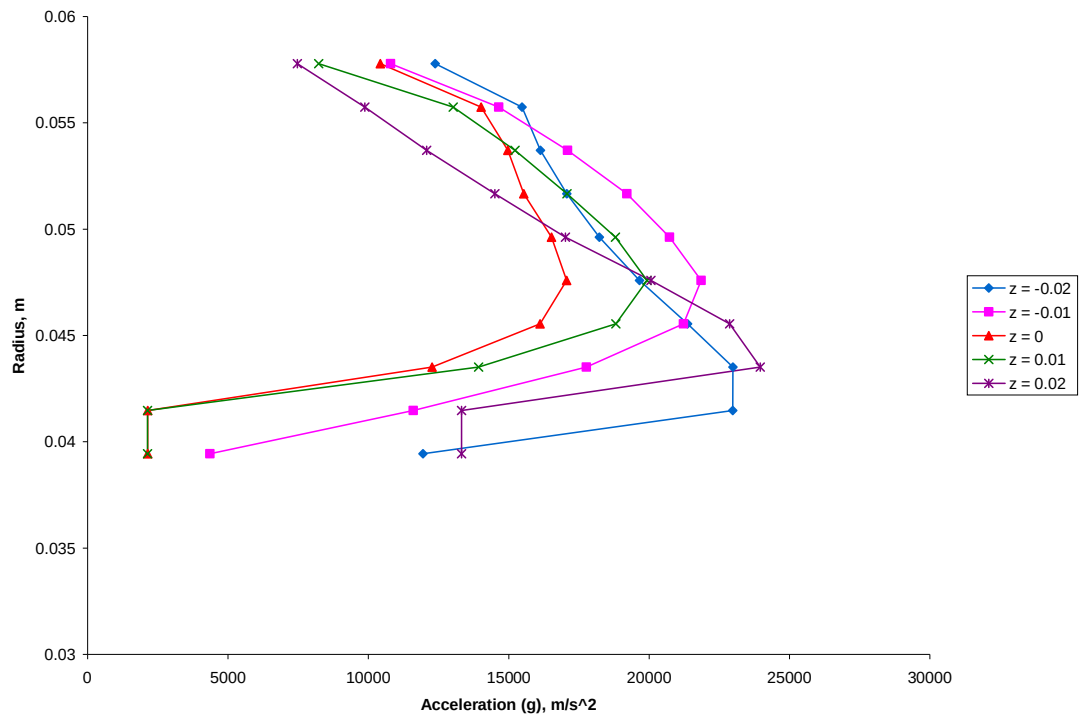


Figure 16 Profiles of Tangential Acceleration within the Circumferential Cavity of the SRV Baseline

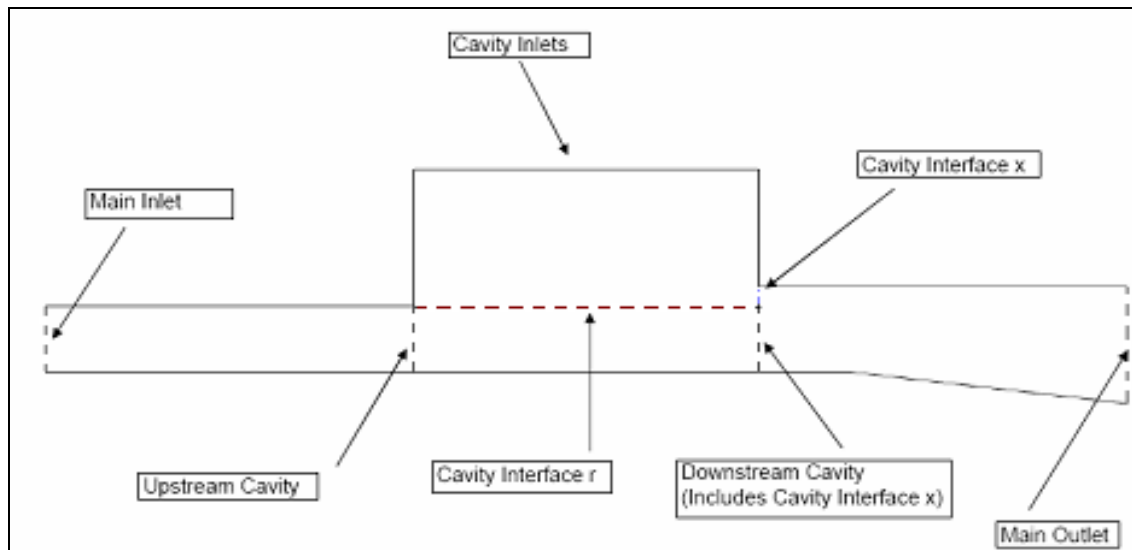


Figure 17 Circumferential Interface Schematic

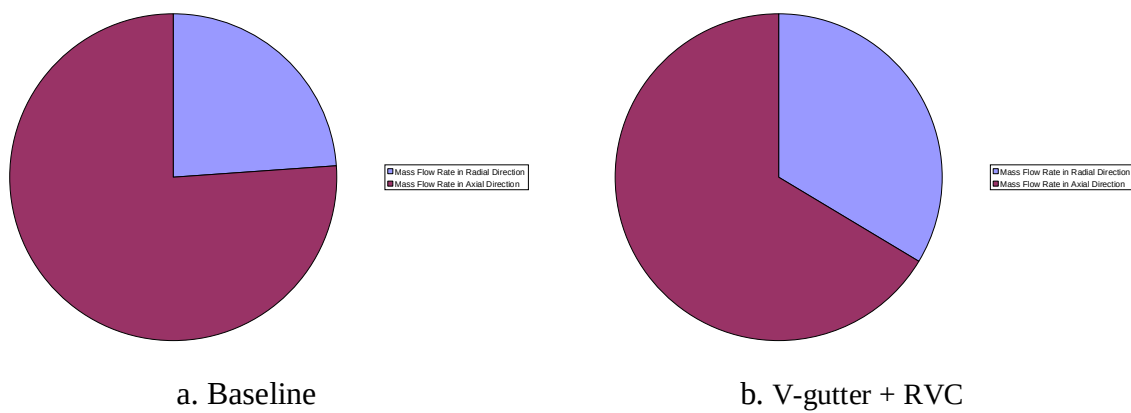


Figure 18 Mass Flow Rate Component Comparison

Appendix II. Parametric Study Figures and Tables

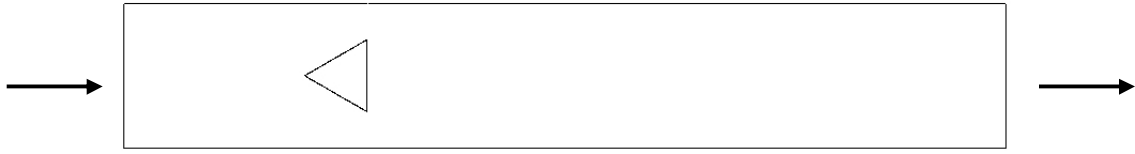
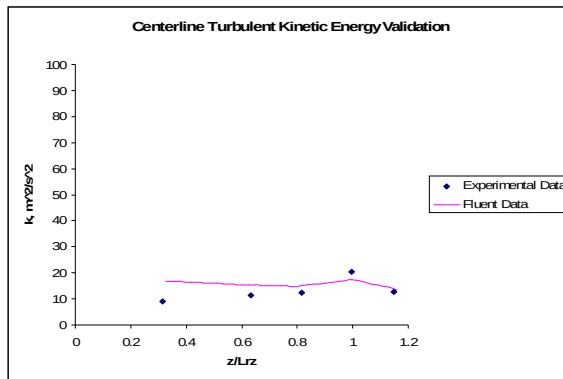


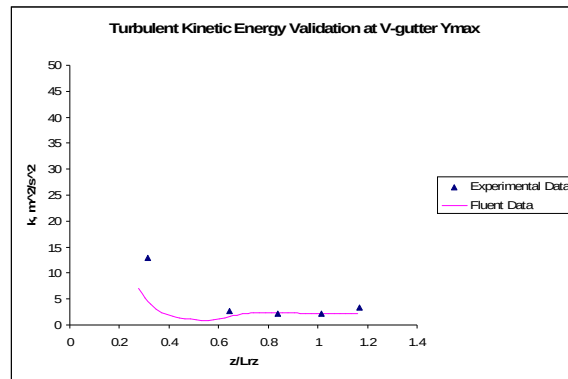
Figure 19 Original Geometry of V-gutter in a cavity

Table 2 Mesh Convergence Study

Grid Size	Exit Temp, K	% diff	Exit Axial Vel, m/s	% diff
20871	1731.41		57.67	
41039	1704.89	1.53	56.18	2.60
82301	1787.43	4.84	60.58	7.83
147051	1812.12	1.38	61.64	1.76
278701	1802.03	0.56	61.22	0.69
361521	1805.10	0.17	61.45	0.38



a. V-Gutter Centerline Comparison



b. V-gutter Y-max Comparison

Figure 20 Validation Results for Turbulent Kinetic Energy extending from the V-gutter Centerline and the V-Gutter Y-max Axially Towards the Outlet [6]

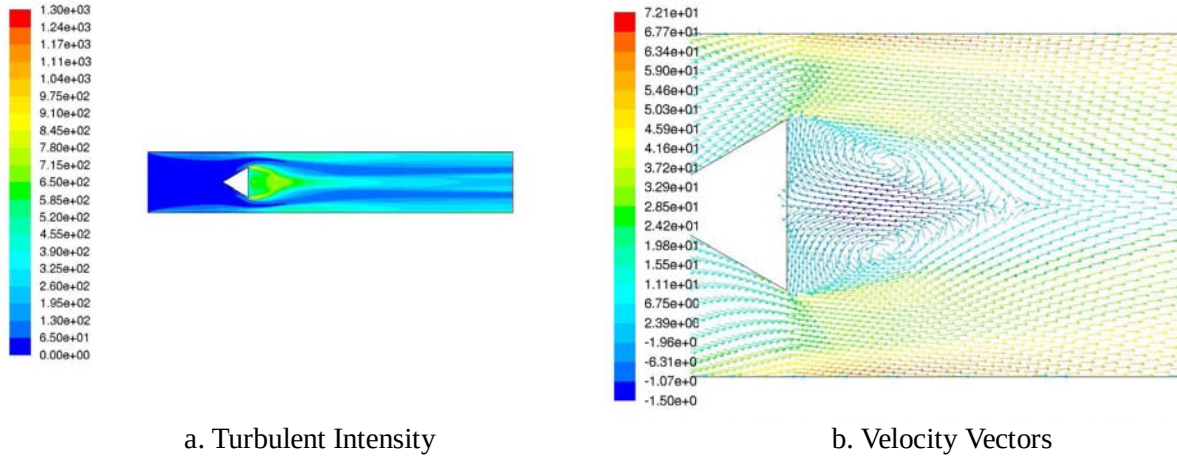


Figure 21 Mixing Properties of Baseline Case Setup

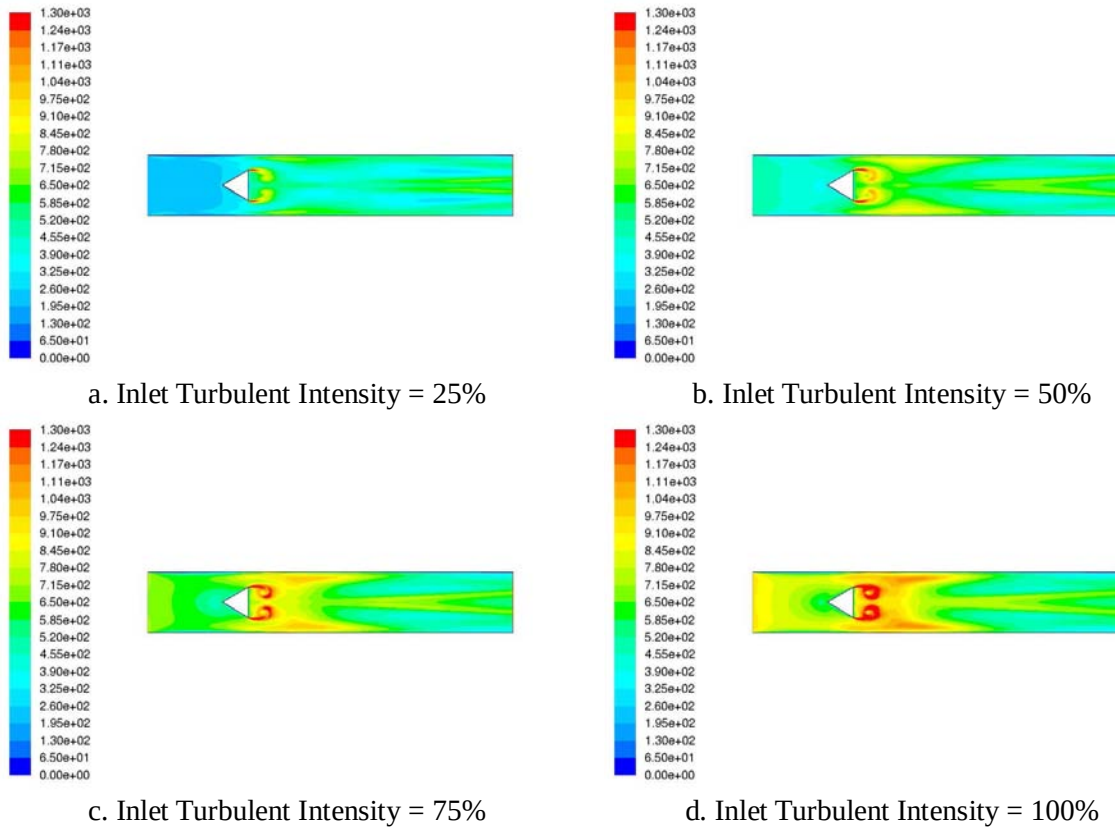


Figure 22 Turbulent Intensity Contours at $\alpha = 0^\circ$ for Four Separate Inlet Turbulent Intensity Conditions (25%, 50%, 75%, and 100%)

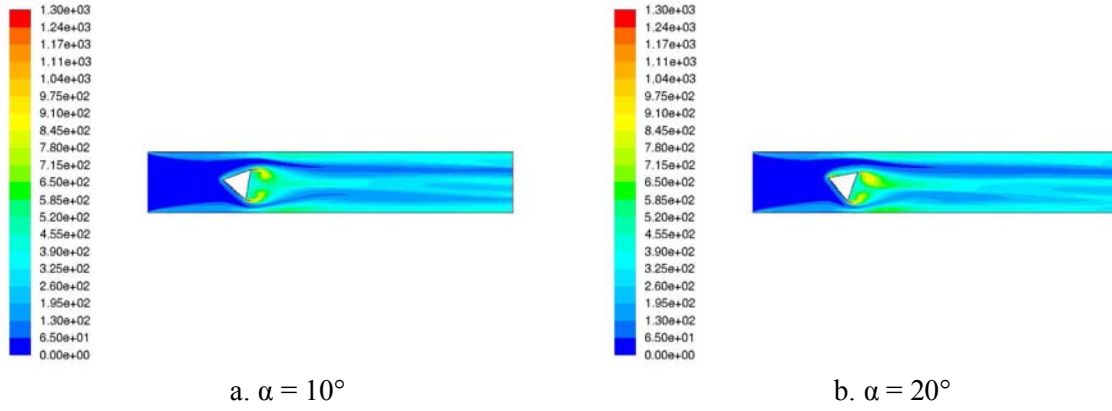


Figure 23 Turbulent Intensity Contours For Two Angles of Attack ($\alpha = 10^\circ$ and $\alpha = 20^\circ$)

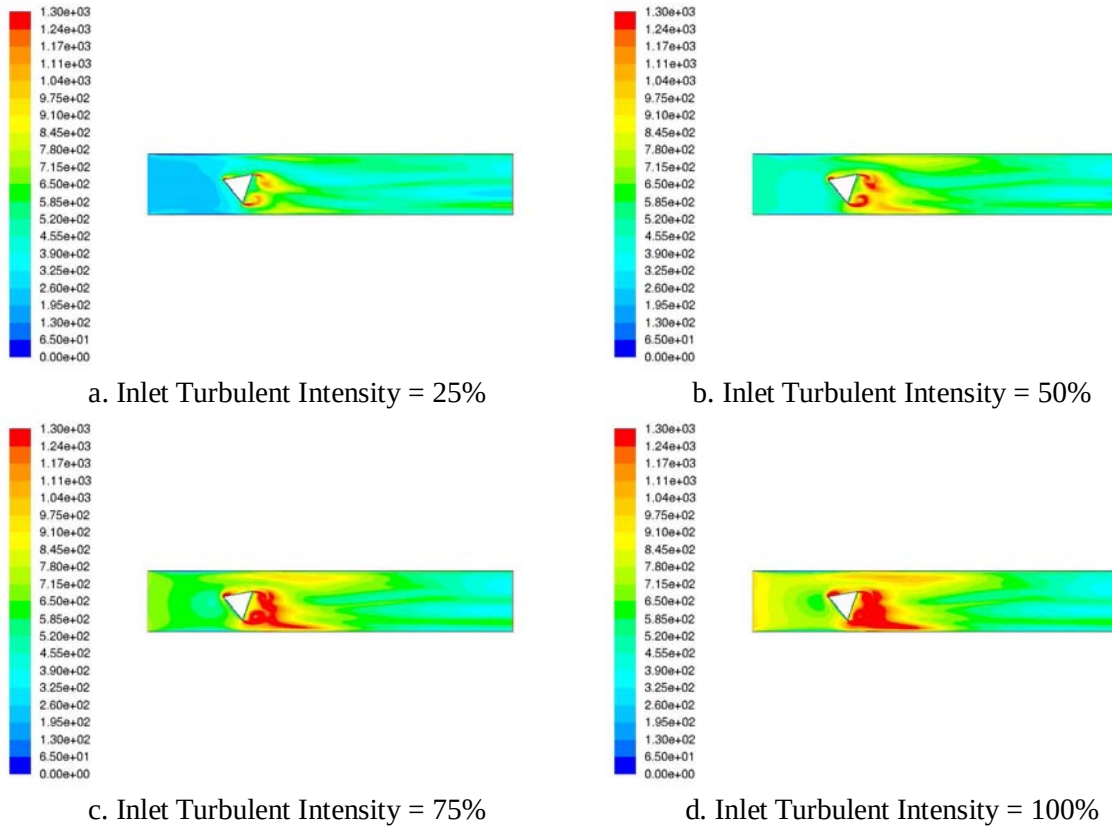
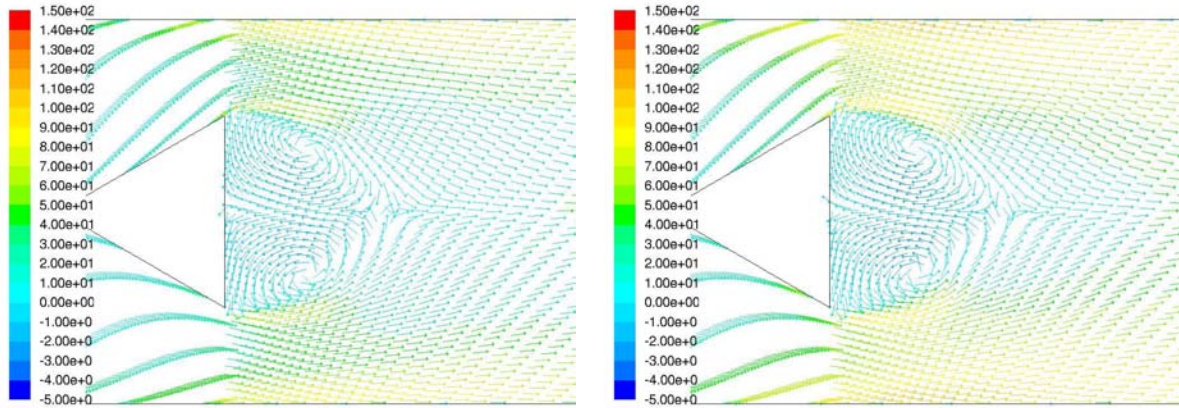
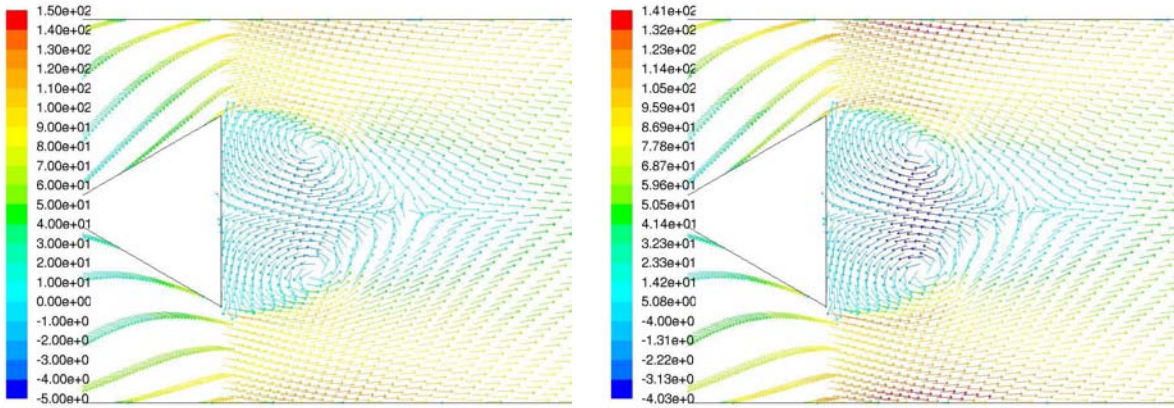


Figure 24 Turbulent Intensity Contours with $\alpha = 20^\circ$ for Four Separate Inlet Turbulent Intensity Conditions (25%, 50%, 75%, and 100%)



a. Inlet Turbulent Intensity = 25%

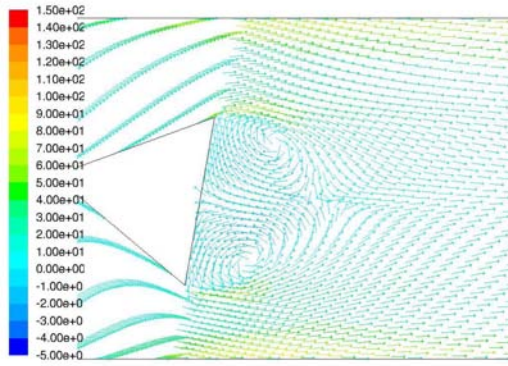
b. Inlet Turbulent Intensity = 50%



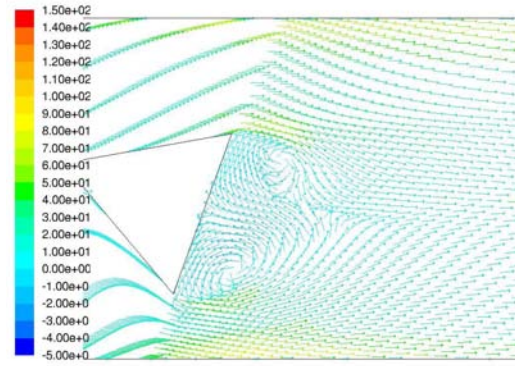
c. Inlet Turbulent Intensity = 75%

d. Inlet Turbulent Intensity = 100%

Figure 25 Velocity Vector Plots Colored by Axial Velocity for Four Separate Inlet Turbulent Intensity Conditions (25%, 50%, 75%, and 100%)

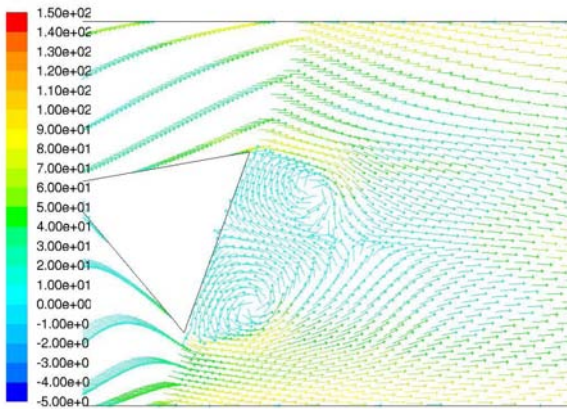


a. $\alpha = 10^\circ$

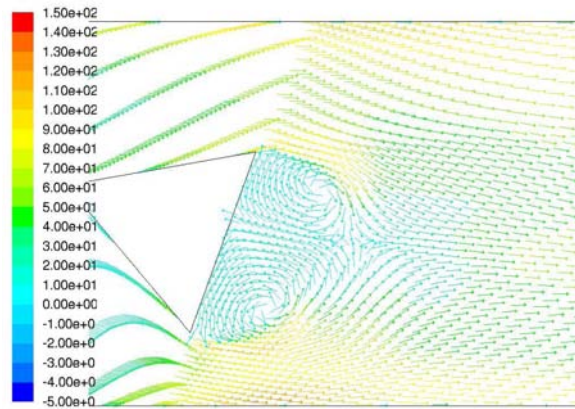


b. $\alpha = 20^\circ$

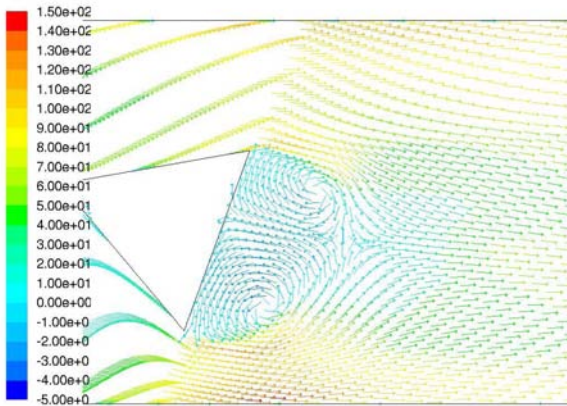
Figure 26 Velocity Vector Plots Colored by Axial Velocity for Two Angles of Attack ($\alpha = 10^\circ$ and $\alpha = 20^\circ$)



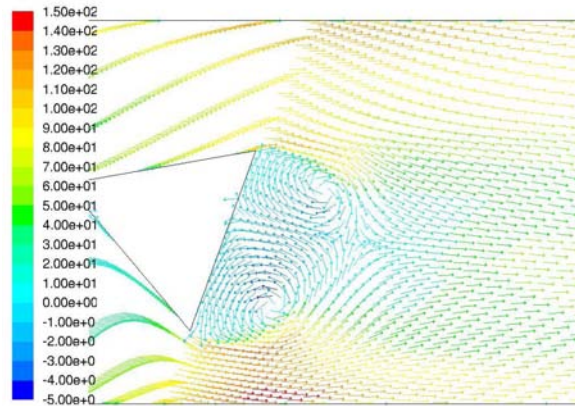
a. Inlet Turbulent Intensity = 25%



b. Inlet Turbulent Intensity = 50%

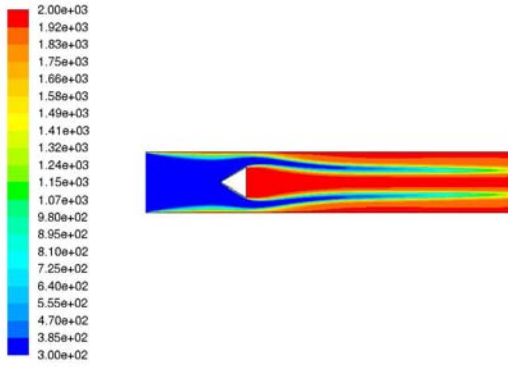


c. Inlet Turbulent Intensity = 75%

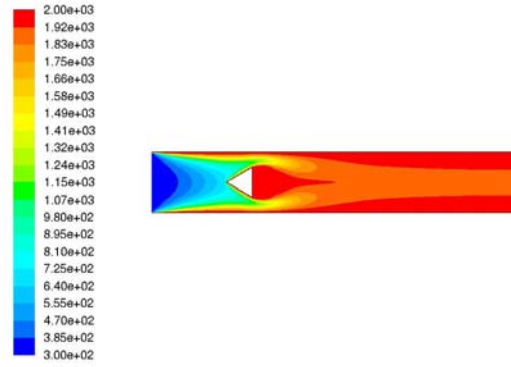


d. Inlet Turbulent Intensity = 100%

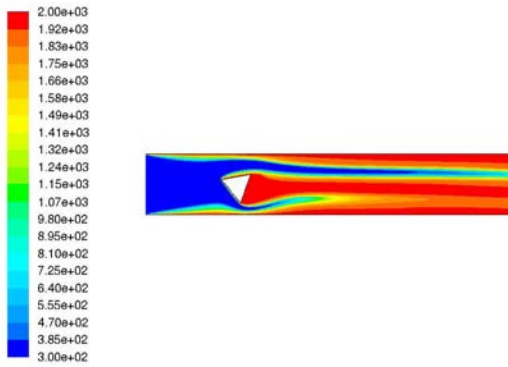
Figure 27 Velocity Vectors Colored by Axial Velocity with $\alpha = 20^\circ$ for Four Separate Inlet Turbulent Intensity Conditions (25%, 50%, 75%, and 100%)



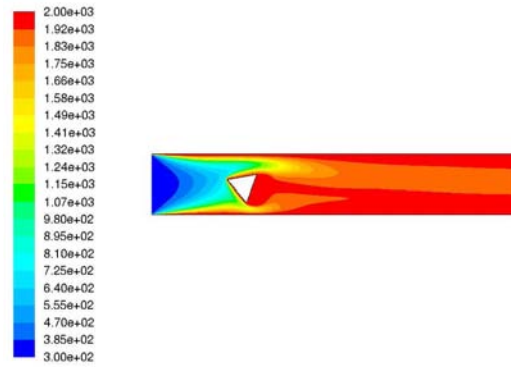
a. Turbulent Intensity = 2%, $\alpha = 0^\circ$



b. Turbulent Intensity = 100%, $\alpha = 0^\circ$



c. Turbulent Intensity = 2%, $\alpha = 20^\circ$



d. Turbulent Intensity = 100%, $\alpha = 20^\circ$

Figure 28 Temperature Contours of Baseline Setup Along with the Most Extreme Case of the Other Three Studies (Turbulent Intensity = 2%, $\alpha = 0^\circ$, Turbulent Intensity = 100%, $\alpha = 0^\circ$, Turbulent Intensity = 2%, $\alpha = 20^\circ$, Turbulent Intensity = 100%, $\alpha = 20^\circ$)

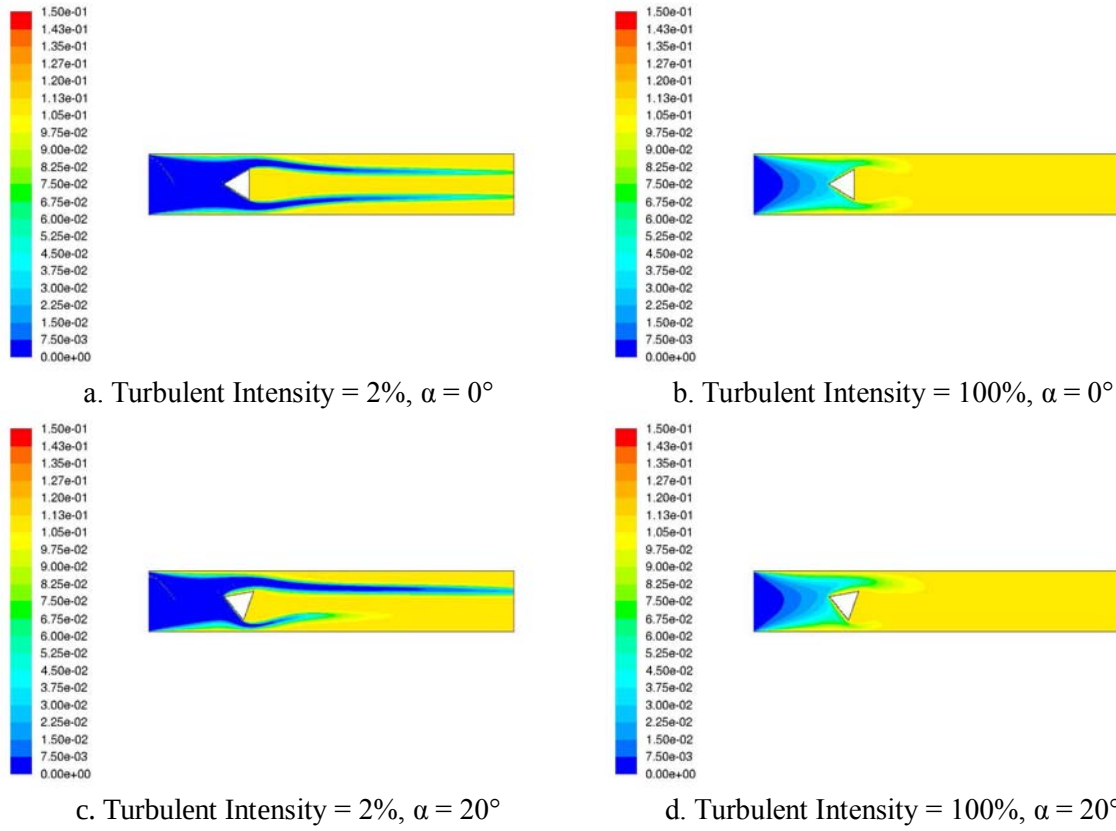
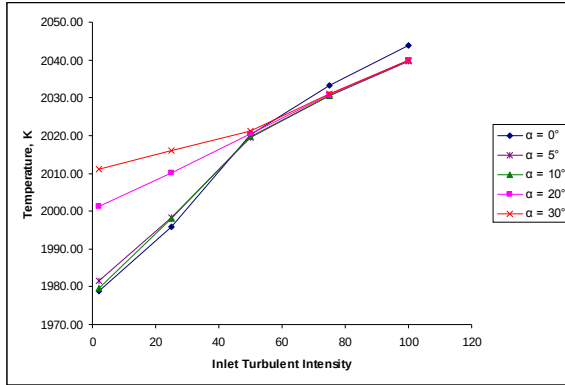


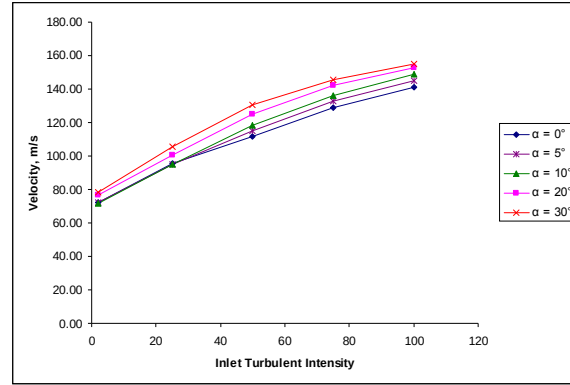
Figure 29 CO2 Contours of Baseline Setup Along with the Most Extreme Case of the Other Three Studies

Table 3 Max Value Comparison of Baseline with the Most Extreme Case of the Other Three Studies

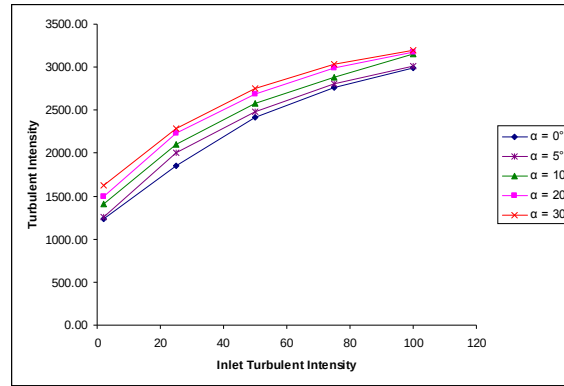
	$\alpha = 0^\circ$, I = 2%	$\alpha = 0^\circ$, I = 100%	$\alpha = 30^\circ$, I = 2%	$\alpha = 30^\circ$, I = 100%
Max Temp (K)	1978.97	2043.76	2011.00	2039.97
Max Axial Vel (m/s)	72.11	141.24	78.43	154.76
Max Turb. Intensity (%)	1235.03	2996.01	1627.61	3200.78



a. Maximum Temperature Values



b. Maximum Axial Velocity Values



c. Maximum Turbulent Intensity Values

Figure 30 Maximum Values of Temperature, Axial Velocity, and Turbulent Intensity

Table 4 Recirculation Zone Size Differences for Varied Inlet Angle of Attack and Turbulent Intensity

	$\alpha = 0^\circ$ I = 2%	$\alpha = 0^\circ$ I = 25%	$\alpha = 0^\circ$ I = 50%	$\alpha = 0^\circ$ I = 75%	$\alpha = 0^\circ$ I = 100%	$\alpha = 30^\circ$ I = 2%	$\alpha = 30^\circ$ I = 25%	$\alpha = 30^\circ$ I = 50%	$\alpha = 30^\circ$ I = 75%	$\alpha = 30^\circ$ I = 100%
X Span	24.8	24.8	24.6	24.5	24.5	22.5	22.1	21.6	20.7	19.9
Y Span										
Top	12.5	12.5	12.5	12.5	12.5	11.4	10.2	8.8	7.9	6.8
Y Span										
Bottom	12.5	12.5	12.5	12.5	12.5	16	17.2	18.7	19.6	20.7
Y Span										
Total	25	25	25	25	25	27.4	27.4	27.5	27.5	27.5

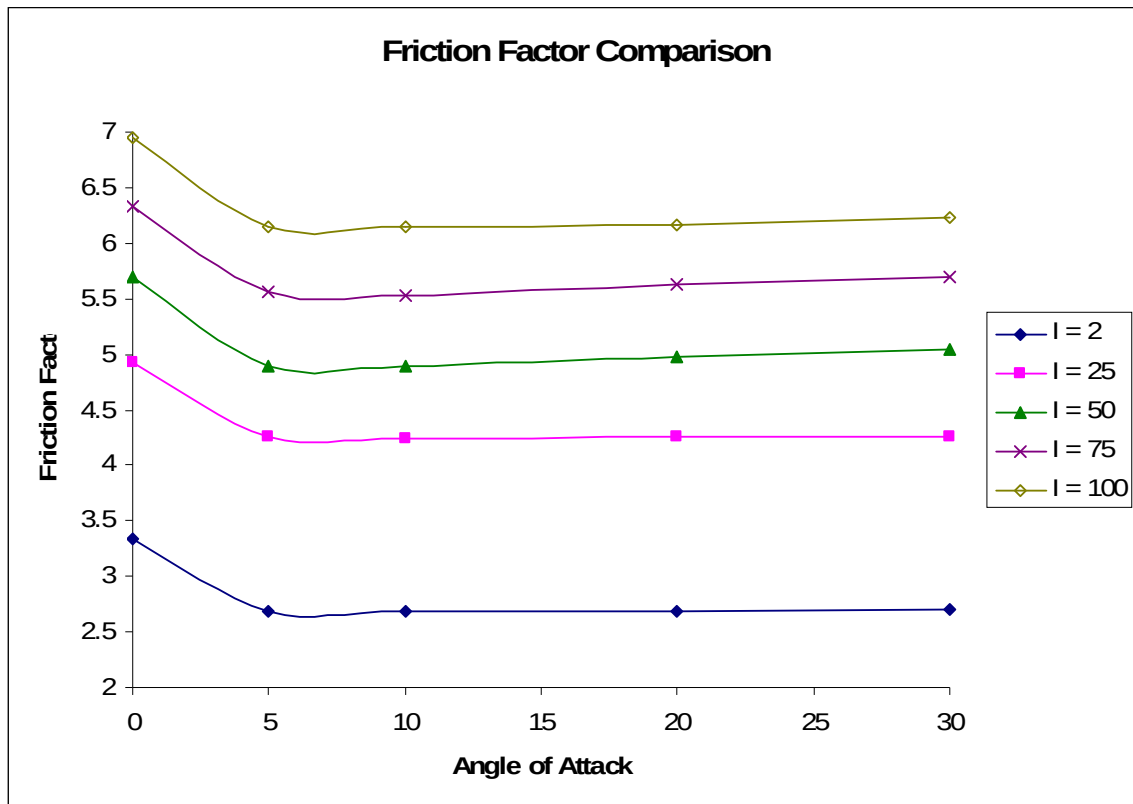


Figure 31 Friction Factor Calculated for Each Case with Lines of Constant Turbulent Intensity Shown

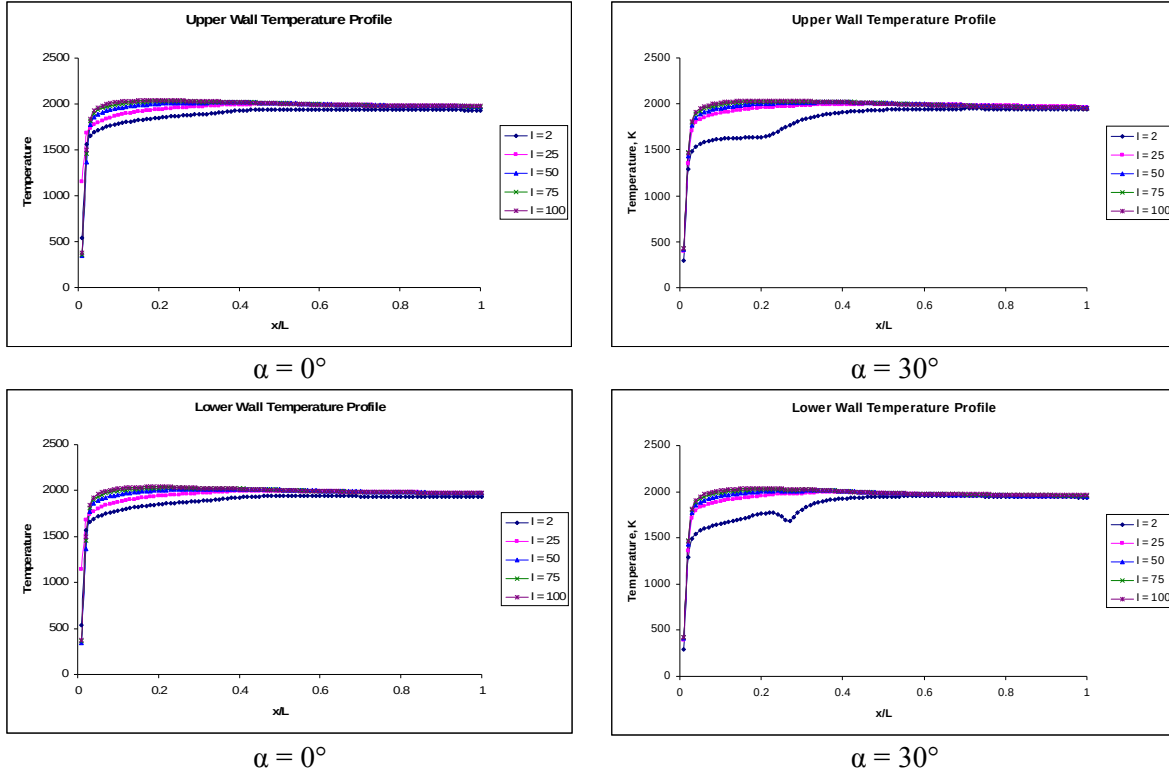


Figure 32 Wall Temperature Profile Comparison of the Upper and Lower Cavity Walls at $\alpha = 0^\circ$ and $\alpha = 30^\circ$

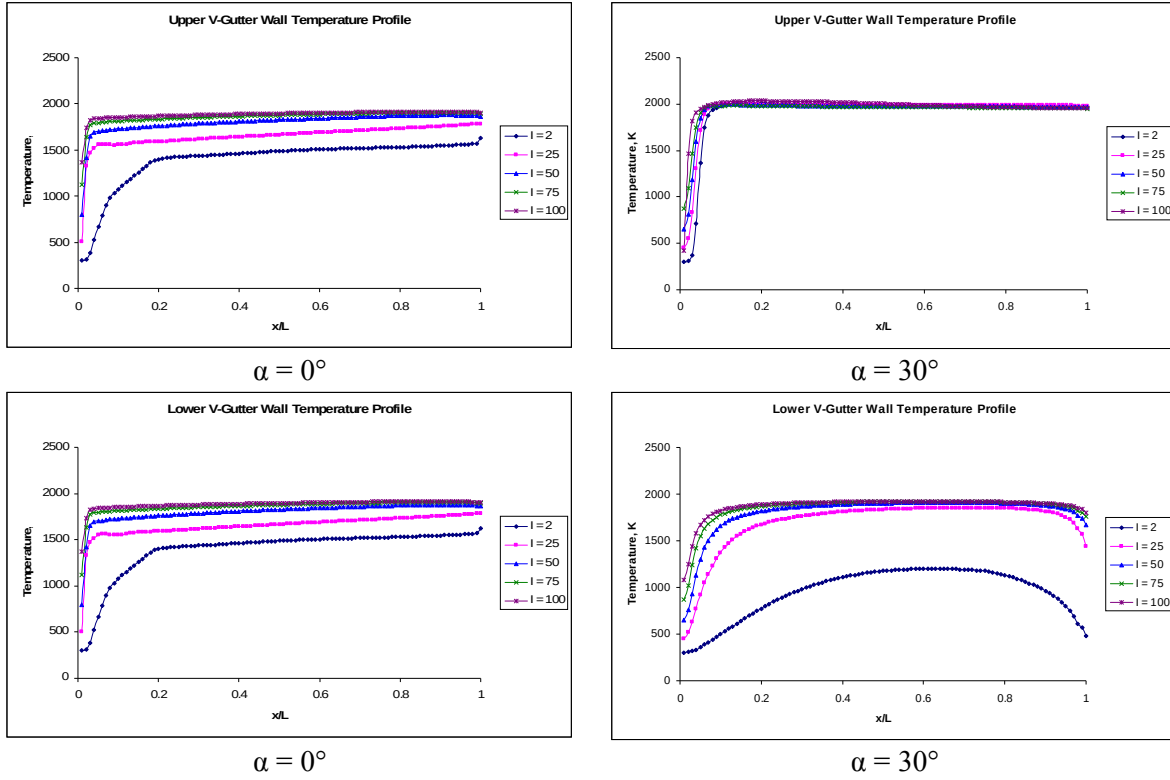


Figure 33 Wall Temperature Profile Comparison of the Upper and Lower V-Gutter Walls at $\alpha = 0^\circ$ and $\alpha = 30^\circ$

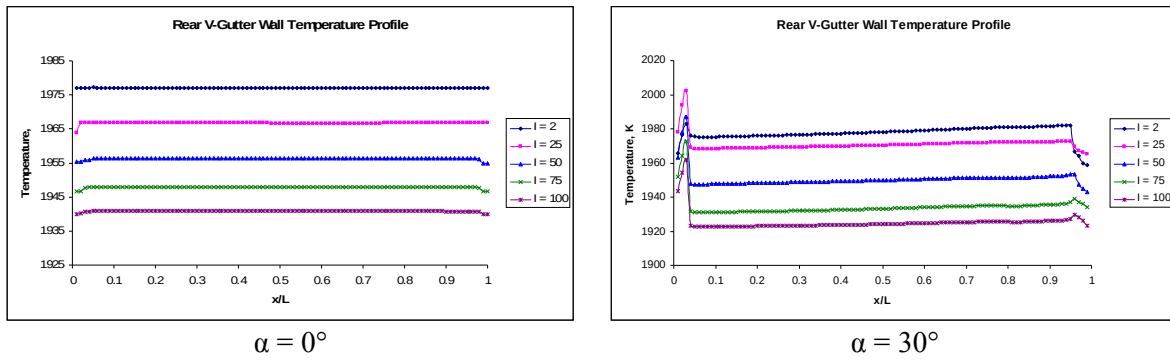


Figure 34 Wall Temperature Profile Comparison of the Rear V-Gutter Wall at $\alpha = 0^\circ$ and $\alpha = 30^\circ$

Appendix III. Comprehensive Baseline SRV ITB Results

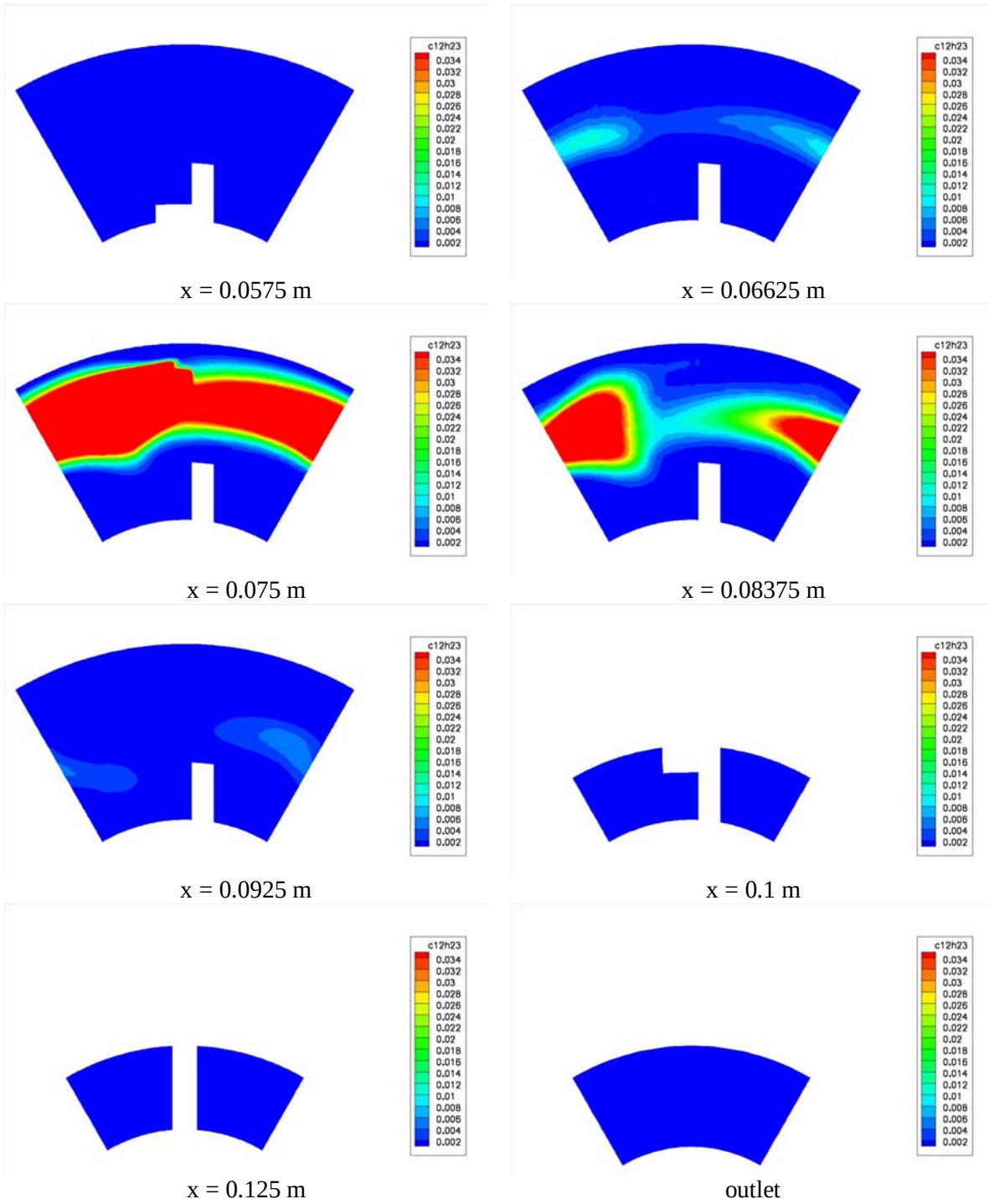


Figure 35 Contours of Propane Concentration for the Baseline Configuration

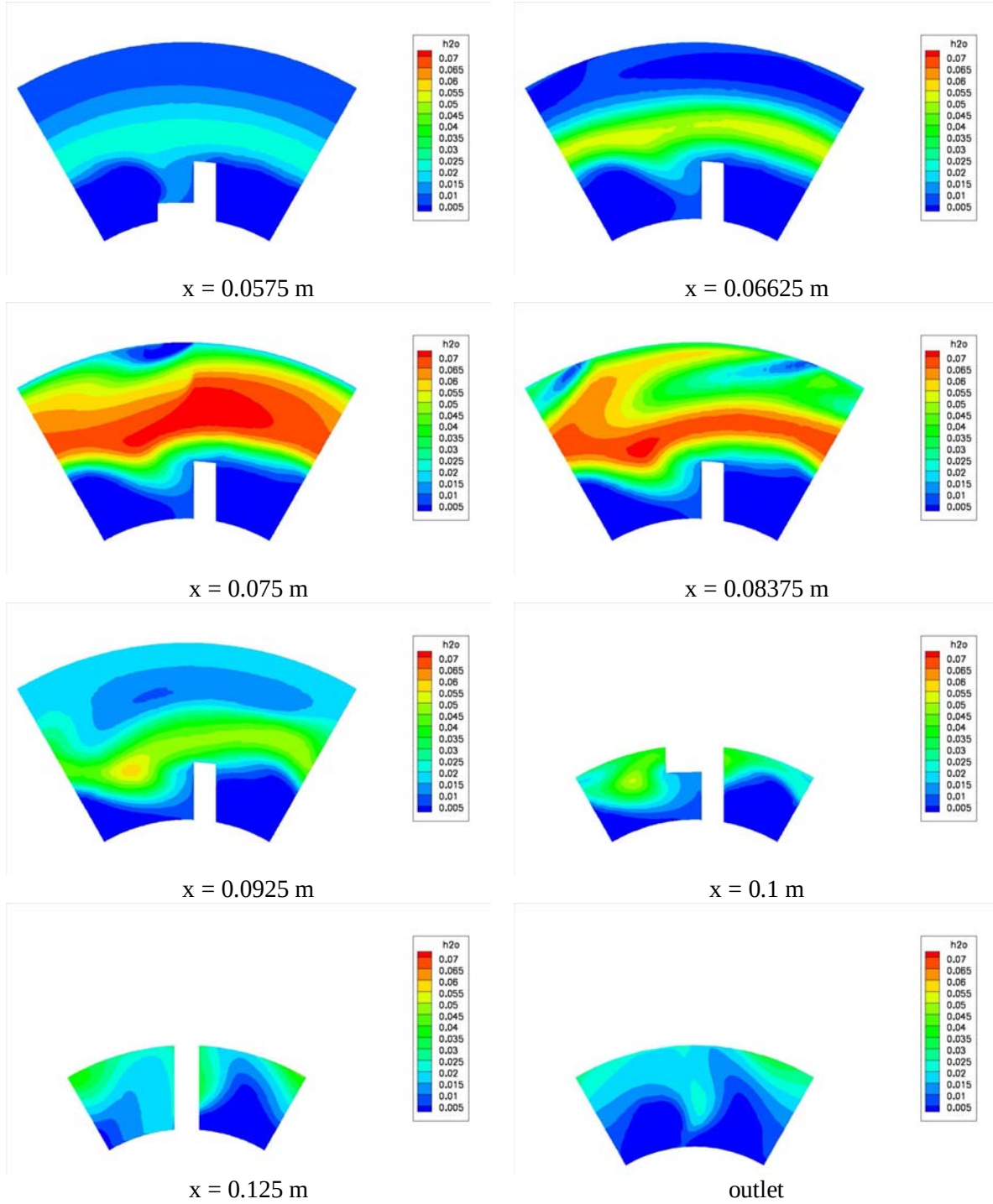


Figure 36 Contours of H₂O for the Baseline Configuration

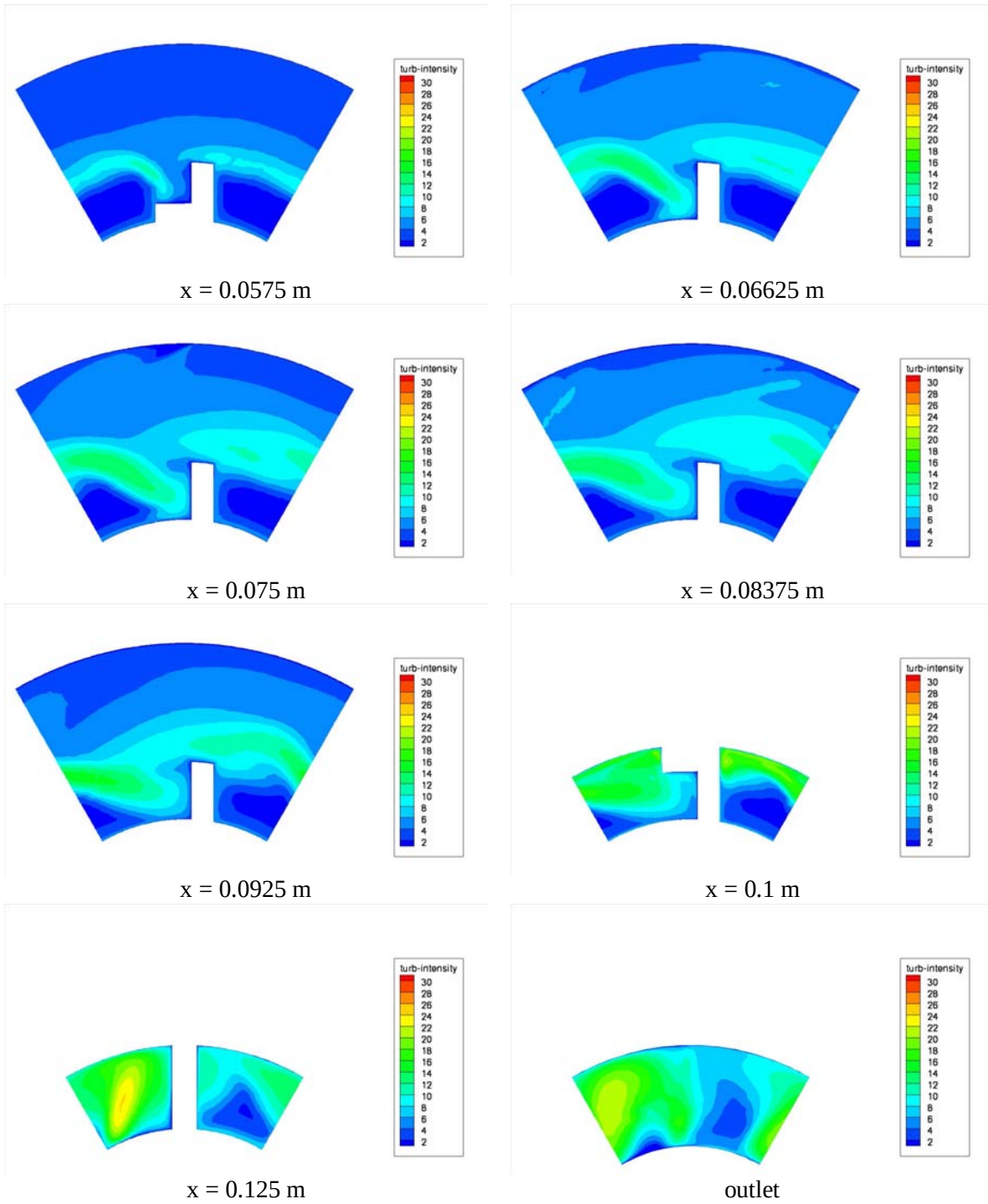


Figure 37 Contours of Turbulent Intensity for the Baseline Configuration

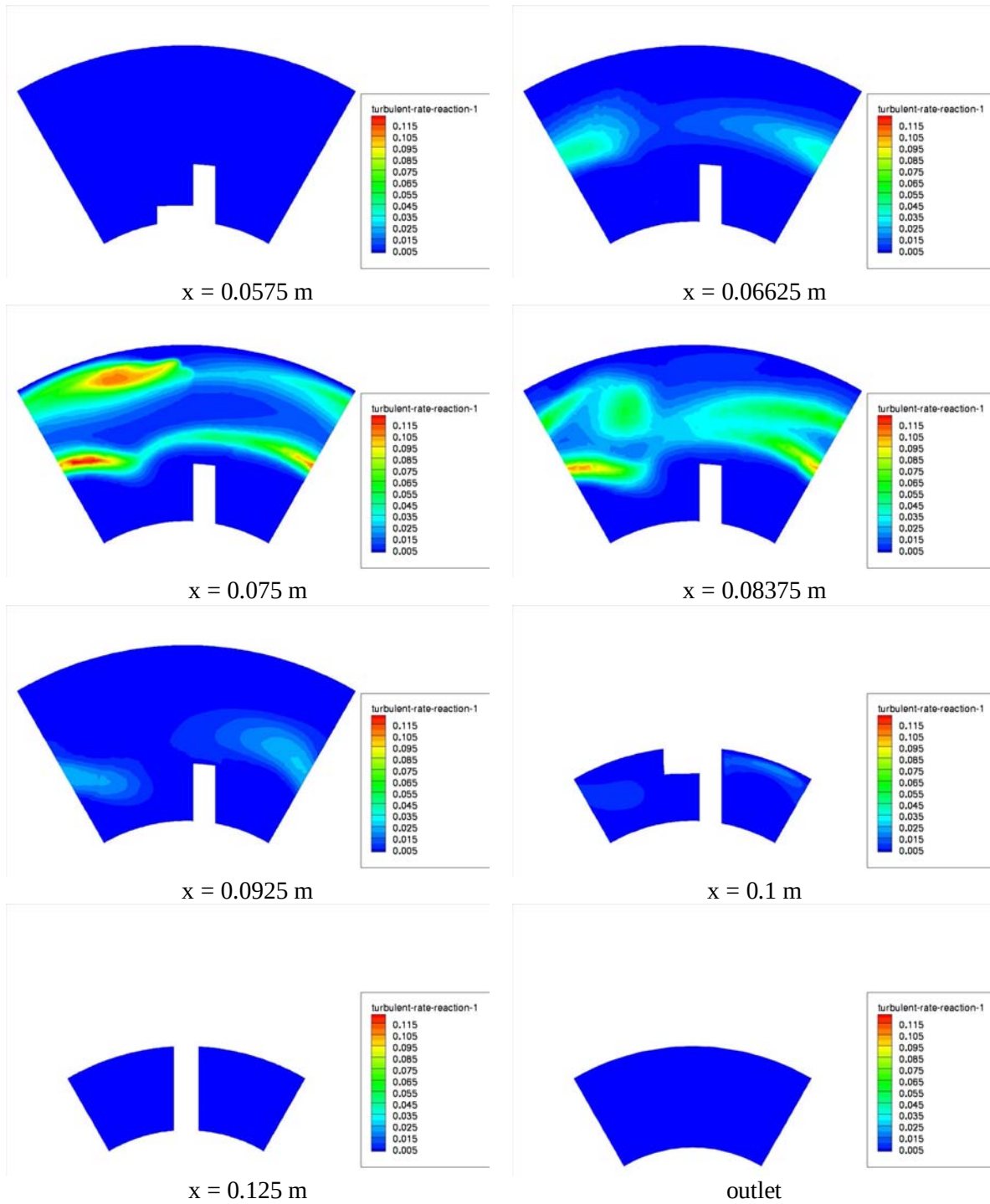


Figure 38 Contours of Rate of Reaction for the Baseline Configuration

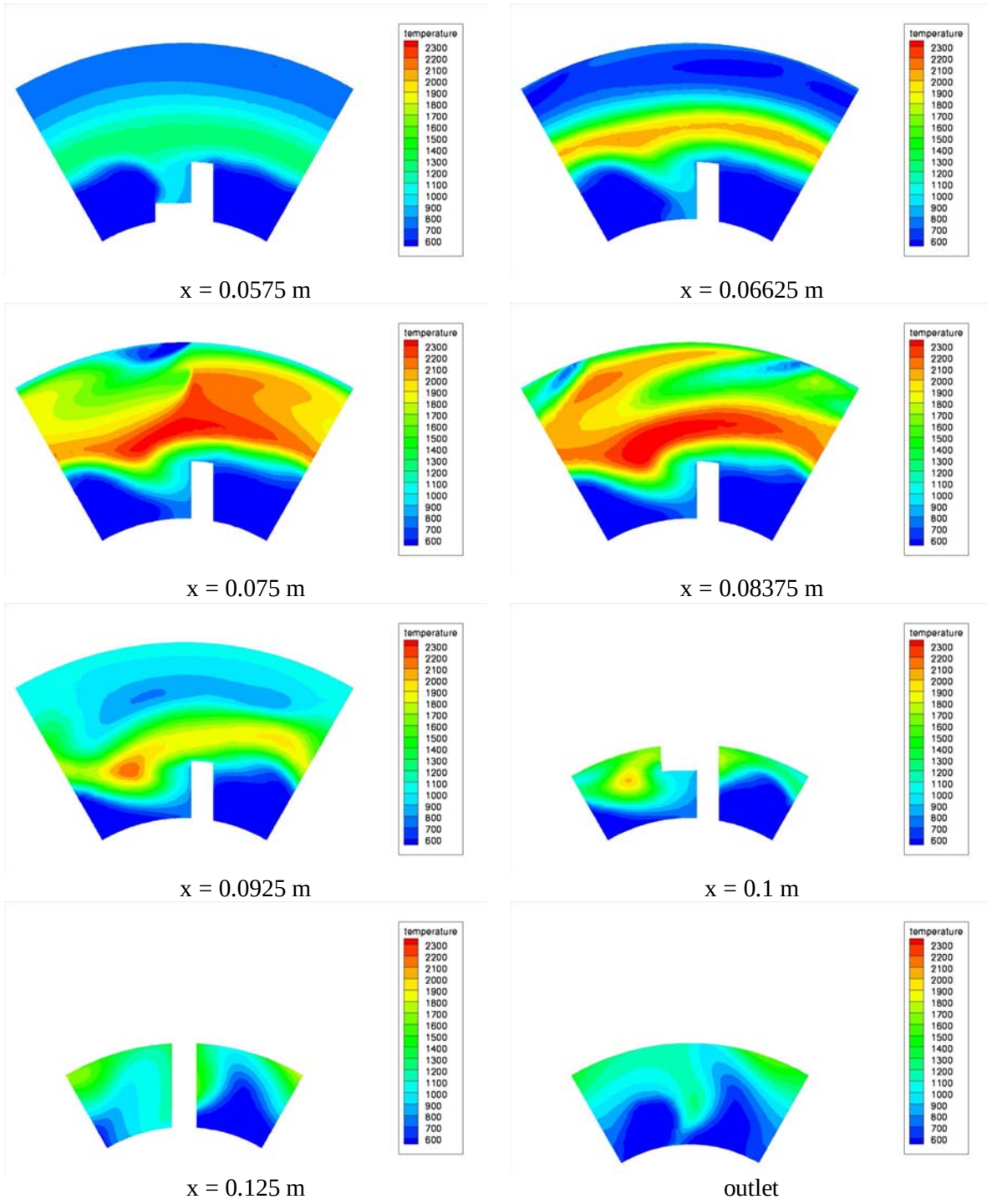


Figure 39 Contours of Temperature for the Baseline Configuration

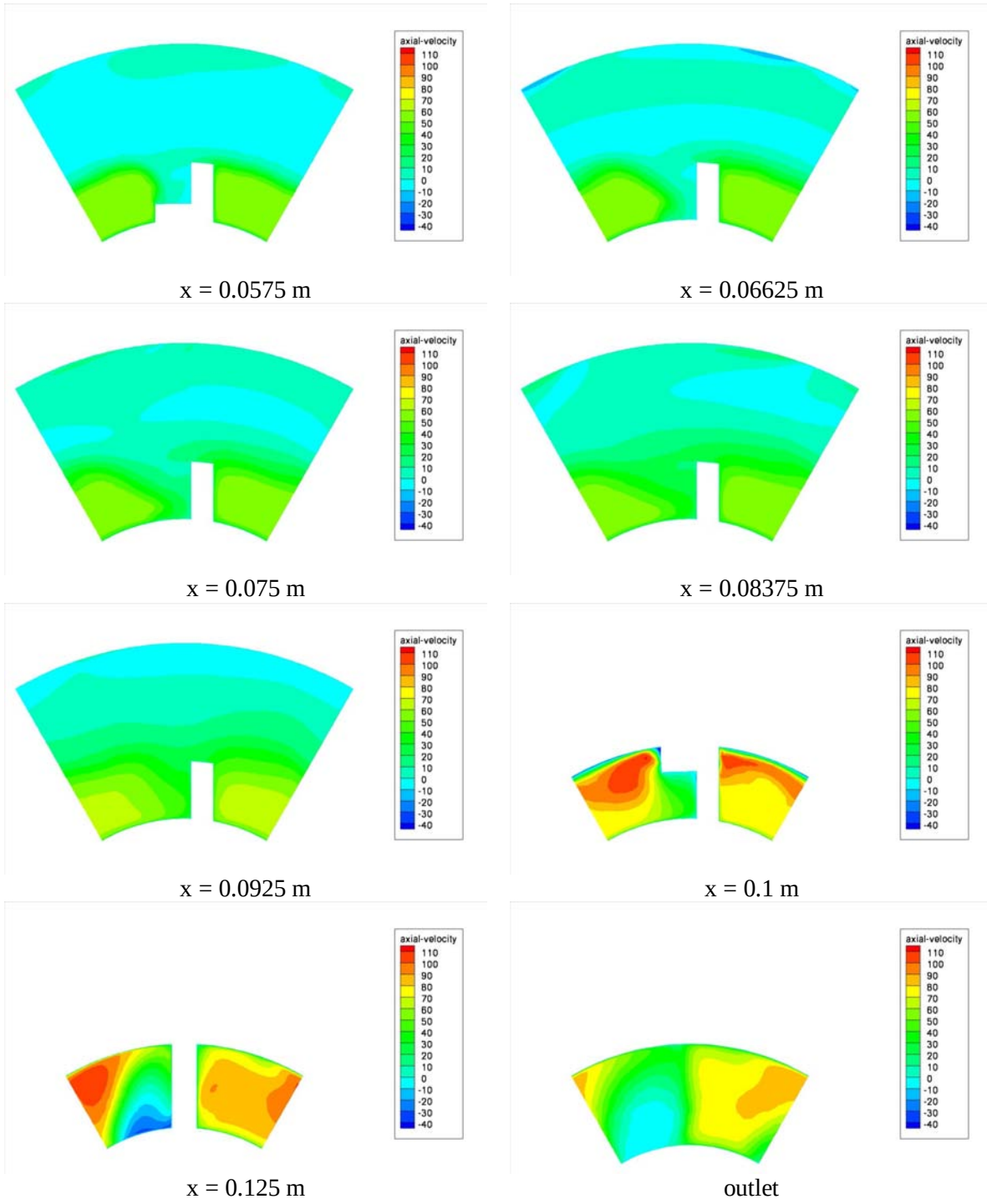


Figure 40 Contours of Axial Velocity for the Baseline Configuration

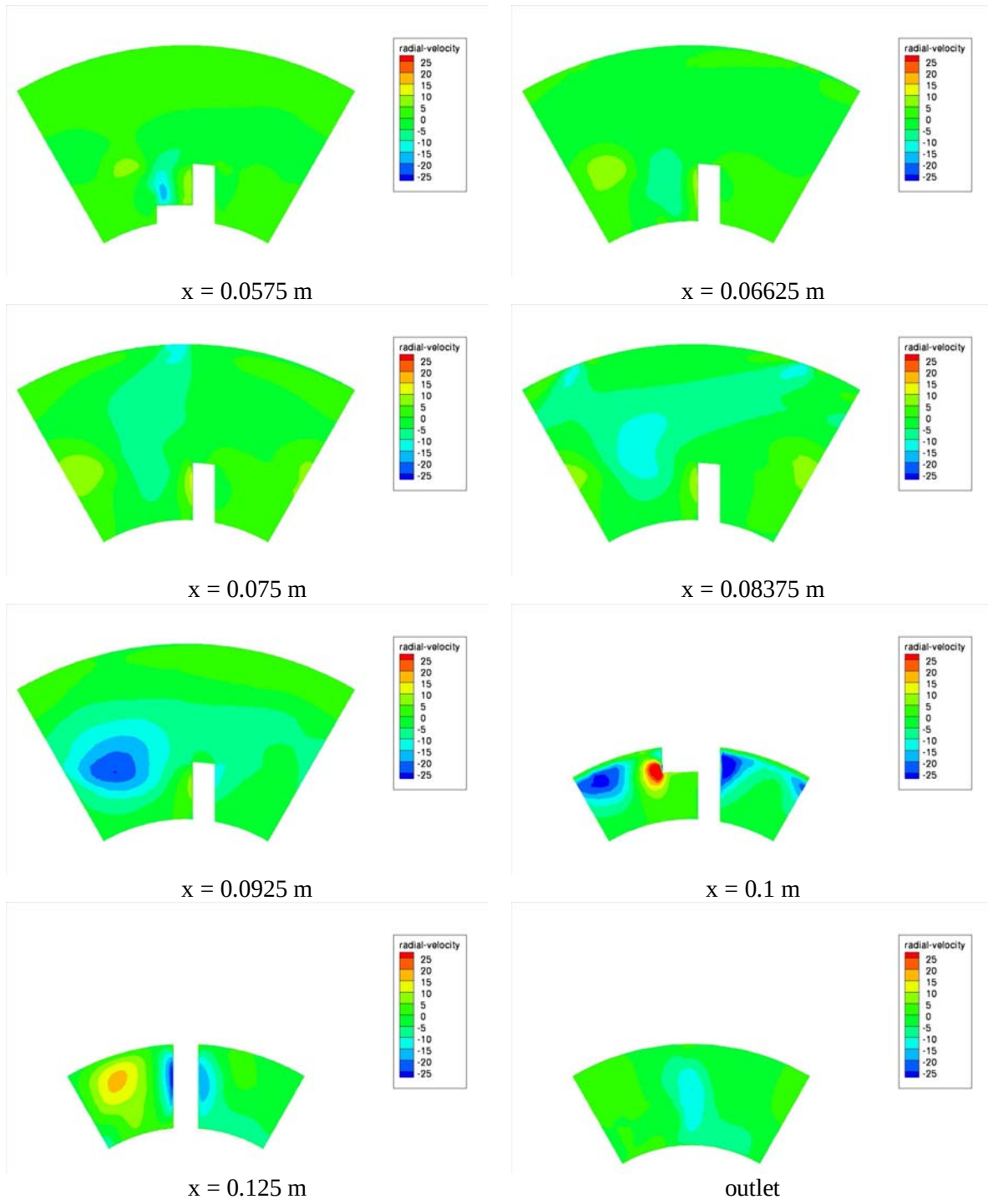


Figure 41 Contours of Radial Velocity for the Baseline Configuration

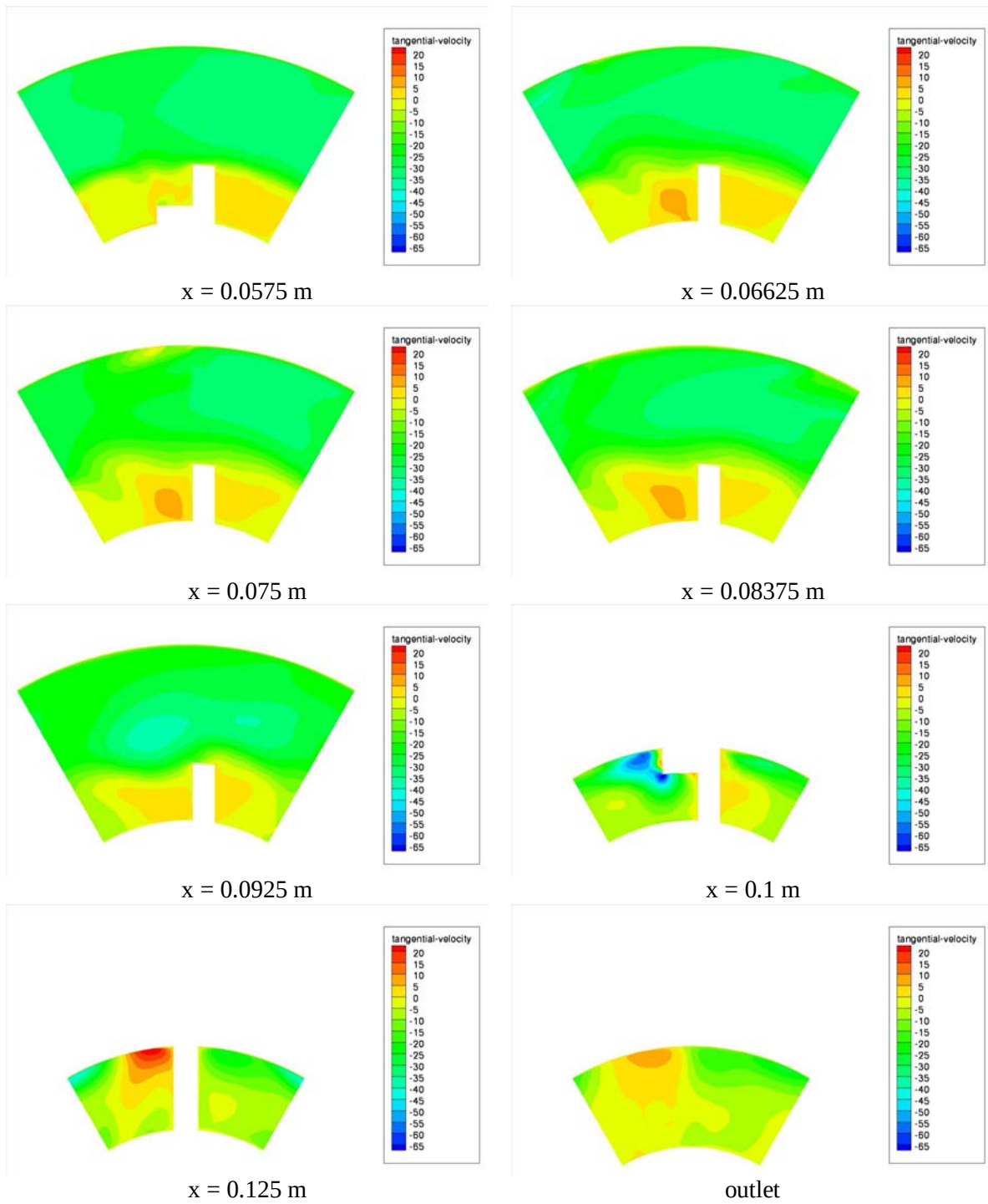


Figure 42 Contours of Tangential Velocity for the Baseline Configuration

Appendix IV. Curved Radial Vane (CRV) ITB

4.1 Introduction

The V-Gutter was also included in the curved vane ITB geometry to explore what effects would occur as a result [15]. Significant benefits had already been seen due to the addition of curved radial vanes into the geometry but at a higher cost of pressure drop. Only one case was run for this configuration, and it was run at a higher equivalence ratio than the previous SRV ITB simulations. The V-Gutter was kept at 0° angle of attack for this simulation. Also, the geometry downstream of the circumferential cavity was extended in order prevent backflow at the exit plane, so all results were examined at the same exit plane as the SRV ITB.

4.2 Boundary Conditions

For the curved vane case, no periodic surfaces were used since there were no periodic regions present in the new domain. This resulted in a larger area which needed to be modeled. Fuel was injected into the cavity through a cone injector with a half angle of 35 degree, a temperature of 308 K, a radius of 0.0014 m, a velocity of 23.18 m/s, and a mass flow of 0.0003882 kg/s per injector with a Rosin-Rammler distribution. The spray model was also set to spray the fuel in 500 separate streams. The air from the main inlet was non-vitiated and released axially through the ITB at a mass flow rate of 0.05708 kg/s. This resulted in a Reynolds number of approximately 14800. Each of the 24 circumferential inlets injects air at a rate of 0.0007686 kg/s. Each mass flow inlet was given a turbulent intensity of 3% and a turbulent length scale of 0.035335 m. At the outlet, the gage pressure was set to zero Pa in order to model the ITB exhausting to atmospheric air. All walls were set to no slip and with zero heat flux. The no slip

condition forces zero velocity tangential to the wall at the surface, which is what causes boundary layers to develop off the walls. The zero heat flux condition forces no heat transfer between the wall and the fluid.

4.3 Convergence

The solution was iterated for at least 3600 times for each case in order to reach convergence. Residual convergence levels were set to 10^{-5} for continuity, momentum, k , and ϵ , 10^{-8} for energy, and 10^{-3} for all other variables. For the first 600 iterations, the Courant number and under relaxation terms for pressure and velocity were set at low values. After 600 iterations, all values were set back to normal (Courant Number = 15, pressure and velocity under relaxation = 0.3). The next 1000 iterations were run with first-order upwind schemes for all variables. Then for the final 2000 iterations, the second order upwind scheme was used for all variables in order to get a converged final solution for each case.

4.4 Computational Mesh

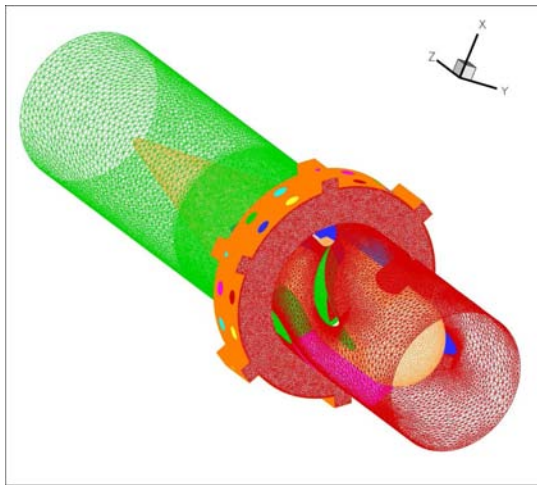
Gridgen V15.10 [24] was first used to create the unstructured surface meshes on the given geometry. The surfaces were loaded into AFLR3D [25], which was used to extrude the volume mesh from the surface meshes. The mesh was then examined to ensure that there was a minimal amount of equiangle and equivolume skew. Special attention was given to the near wall grid distribution as enhanced wall functions were utilized in this study. A near wall spacing was used which resulted in y^+ values between 1 and 7, with the large majority of the values being less than 5. The smaller near wall spacing was necessary to properly capture the interaction of the fluid with the surface of the flameholder and the resulting viscous boundary layer. Also, the

flow was anticipated to be more complex, especially in the areas where the flameholder intersected the radial vanes. The final mesh for the curved vane ITB was approximately 7.5 million cells. Visualization of the mesh can be seen in Fig. 43.

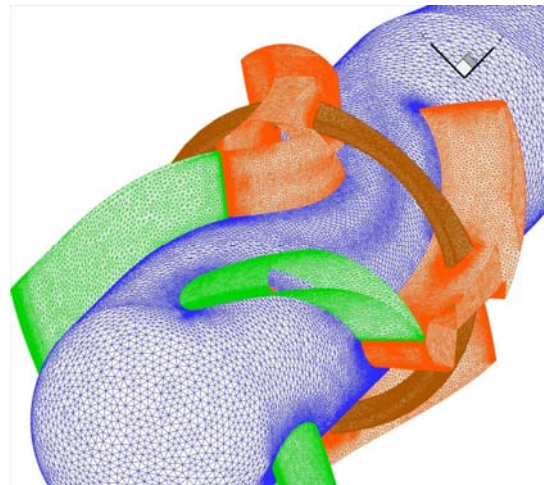
4.5 Results and Discussion

Fig. 44 shows a comparison of the temperature contours at the outlet for the curved vane ITB with no V-Gutters and the same geometry and boundary conditions. A large difference can be seen between the two. The areas of high temperature move from the outer wall of the ITB to the hub of the ITB with the inclusion of the V-Gutter. This causes a significant reduction of temperature around the outer wall of the ITB for the V-Gutter case, but it appears that the areas of high temperature may be more poorly distributed than for the CRV case with no V-Gutter.

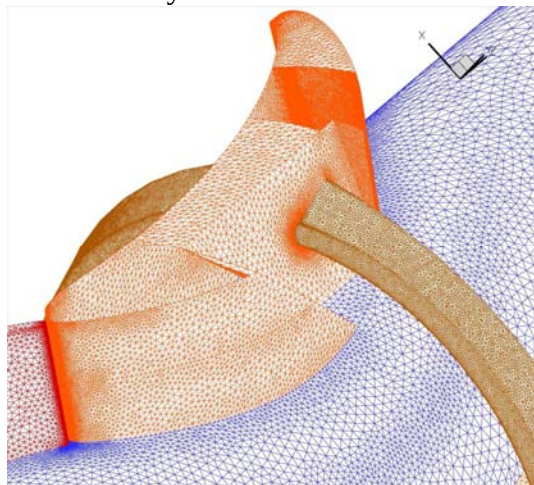
Mass averaged temperature profiles at the exit plane of the two cases can be seen in Fig. 45. These results verify what was suspected from the previous figure. It can be seen that the temperature profile for the CRV with no V-Gutter peaks at the outer and inner walls of the ITB. These results are not ideal, so some improvement could be made. The CRV with V-Gutter, however, gives and even worse exit temperature profile. The profile peaks at the outer wall of the ITB at a significantly higher value than the other case.



a. Full Curved Vane ITB Geometry



b. View of Hub with Vanes and V-Gutters



c. Close up of CRV with the Intersecting V-Gutter

Figure 43 Computational Mesh for Curved Vane ITB with V-Gutter

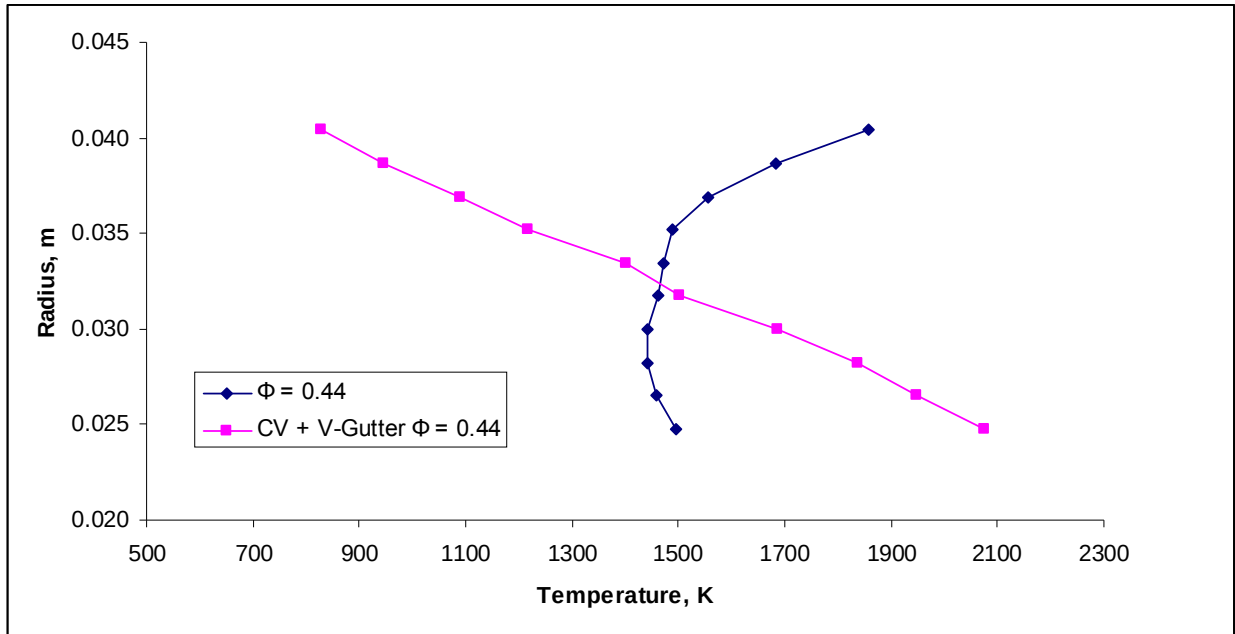


Figure 44 Mass Averaged Temperature Profile for the Curved Vane ITB With and Without V-Gutters

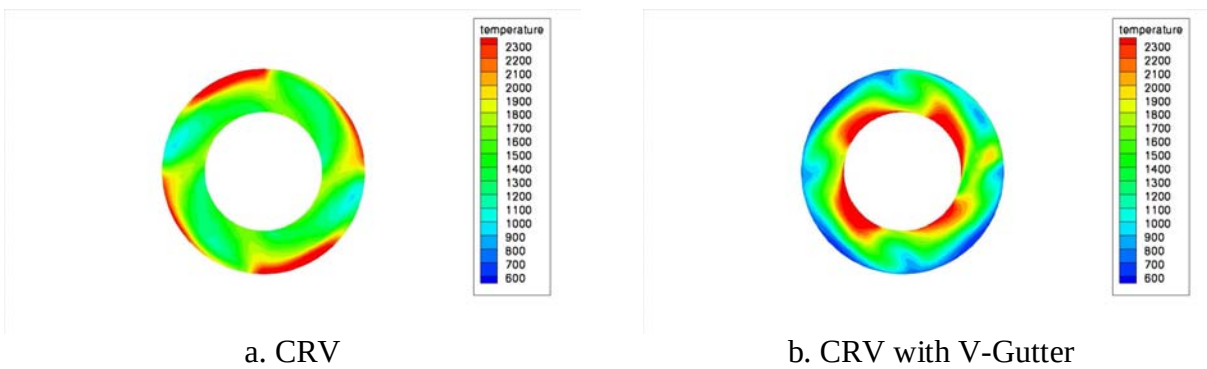


Figure 45 Temperature Contours of CRV ITB With and Without V-Gutters

4.6 Conclusion

It appears that the inclusion of a V-Gutter in the curved vane geometry actual degrades the quality of the parameters of interest at the outlet. However, these are just preliminary results, and more post processing still needs to be performed. Further research needs to be performed in this area, as well. Work which has been performed since this study has shown that the optimal equivalence ratio range for the curved vane ITB is much smaller than what is studied here. The V-Gutter may provide more benefit for these cases.

Vita

Richard Holder was born in Knoxville, TN on June 18, 1983. He was raised in Nashville, TN, where his education began at Crieve Hall Elementary School and continued at Christ the King School. He attended high school at Father Ryan High School. After that, he moved to Knoxville, TN and enrolled at the University of Tennessee, Knoxville, where he attained his Bachelor's of Science in Aerospace Engineering. He went on to graduate school at the University of Tennessee, Knoxville as well. In graduate school he worked as a graduate assistant for Dr. Cheng-Xian Lin, who was also his thesis advisor. Through his assistantship, he learned all of the computational fluid dynamics theory and tools demonstrated in this thesis. This work was done both at the University of Tennessee, Knoxville, and at Wright Patterson Air Force Base where Richard spent a summer working with Dr. Lin. With the completion of this thesis, he will complete his Master's of Science in Mechanical Engineering.